Introduction to Benchtop Equipment and Data Acquisition

Introduction:

Many sensors produce analog (continuous) electrical signals. Once recorded, these signals are scaled to determine the magnitude of the physical phenomenon. In this lab, you will learn about some common methods for recording and displaying voltage signals. Your focus will not be on any particular sensing technology. In the process of performing the lab exercises, you will learn about hardware and software that you will use throughout the semester.

Teaching Objectives:

- Learn to use a function generator to produce a variety of voltage signals.
- Become familiar with an oscilloscope and use it to record and analyze voltage signals.
- Become familiar with the lab PCs and PC-based data acquisition hardware.
- Learn how to write a simple LabVIEW VI to read a voltage signal and write data to a file.

Procedure:

Part 1: Creating a Signal with the Function Generator

Throughout this lab you'll be asked to create different signals with the function generator. The function generator at your workstation. Can create periodic square waves, sine waves and triangle waves. These waveforms can be injected into a circuit under test and analyzed as the waveform progresses to confirm the proper operation of the device or to pinpoint a fault. Try to be accurate, but don't waste a lot of time trying to get the "exact" frequency or voltage levels listed. The purpose of this lab does not depend on meeting exact values; rather, this lab is to help you become familiar with how the signal characteristics can be manipulated.

1. Turn on the oscilloscope and the function generator.

2. Use the frequency range buttons along the top of the function generator to produce a sine wave with a frequency in the range of 200 to 500 Hz.

3. Use the frequency knob (bottom left) to adjust the frequency.

4. Adjust the signal amplitude and DC offset.

Press the Display Select button next to the LCD readout.

Turn the appropriate knobs located on the right side of the function generator (above the BNC connector).

5. Using a BNC cable, connect the MAIN OUT of the function generator to CH 1 of the oscilloscope.

6. Use the CH 1 and CH 2 buttons on the o-scope to turn the channels On or Off (e.g. to turn off CH 2, press CH 2 and then press OFF).

7. Push the AUTOSET button. This will set the triggering, horizontal and vertical so that the signal is displayed nicely on the screen.

Part 2: Analyzing a Signal with an Oscilloscope

The digital oscilloscope (o-scope) is used for viewing voltage signals that change with time. The o-scope measures voltage and displays thousands of samples over time. It is easier to visualize the characteristics of a signal with an oscilloscope than with a digital multimeter.

2.1 Oscilloscope Menu

Press the MENU button to bring up the menu options for the active channel. The far left option is the Coupling button.

- AC coupling only displays the AC (or changing) components of a wave.
- We will typically use DC coupling, which displays both DC (constant) and AC (changing) components.
- The Ground coupling option shows the ground level (0 volts) for channel 1.

When finished with the MENU: push the MENU button, followed by the Off button.

2.2 Setting the Vertical Sweep

The vertical SCALE knob (under the VERTICAL heading) is used to adjust the vertical range of the oscilloscope trace. The number next to CH1 in the lower left corner of the LCD screen shows how many volts are represented by each horizontal line on the screen.

- Turn the SCALE knob and note that the highlighted value changes. (To determine the amplitude of a signal, it is necessary to know how many volts each division represents.)
- Adjust the VERTICAL POSITION knob to move the entire trace up and down. The number 1 and the right arrow on the left side of the display indicate the ground level for channel 1.
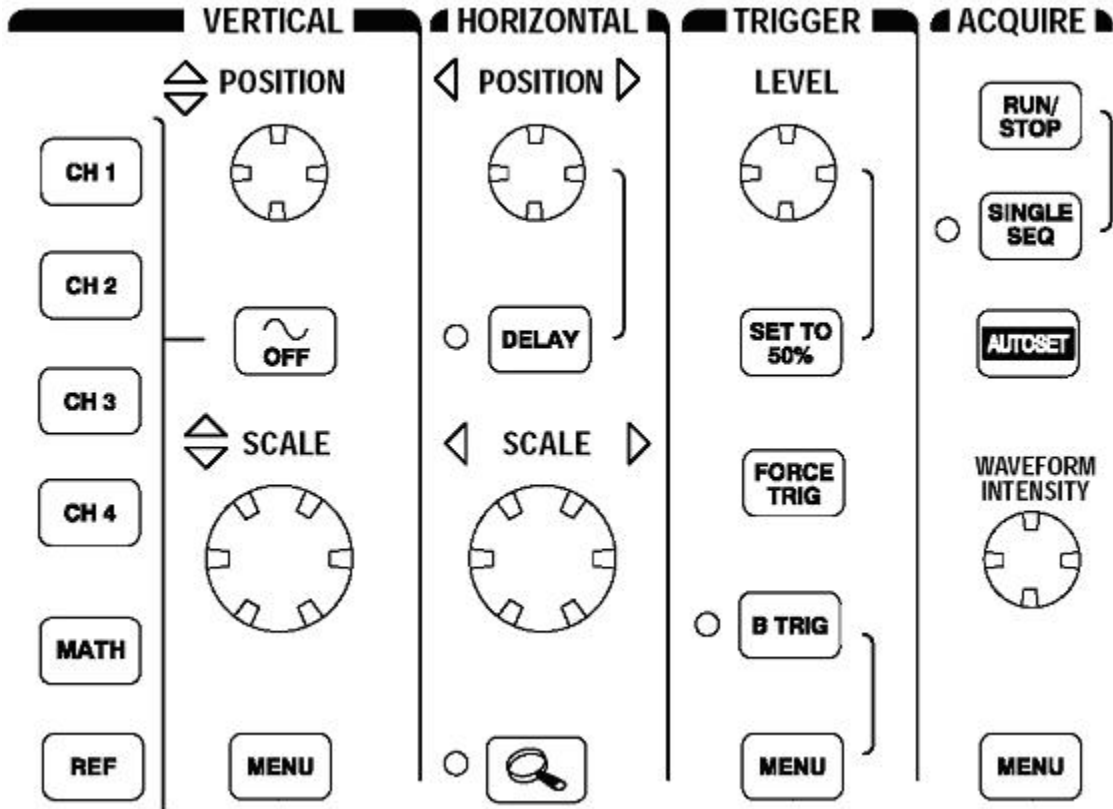
Oscilloscope adjustments

2.3 Setting the Horizontal Sweep (refer to Figure 1)

The SCALE knob under the HORIZONTAL heading is used to set the horizontal sweep speed.

- Turn this SCALE knob and observe how it changes the appearance of the signal on the o-scope.
- The number of seconds (usually milliseconds or microseconds) between each vertical line on the LCD screen is shown at the bottom center of the screen.

The HORIZONTAL POSITION adjustment moves the trace horizontally.

2.4 MEASURE features (refer to Figure 2)

- Push the MEASURE button to bring up the MEASURE menu.
- Select the Select Measurement for Ch1.
- Scroll through the new menu that appears on the right side of the screen. Measure the frequency, period, and mean value of your input signal. Compare these measured values with the settings of the function generator.

Turn off the measurements you selected by pushing the Remove Measurement button from the MEASURE menu.
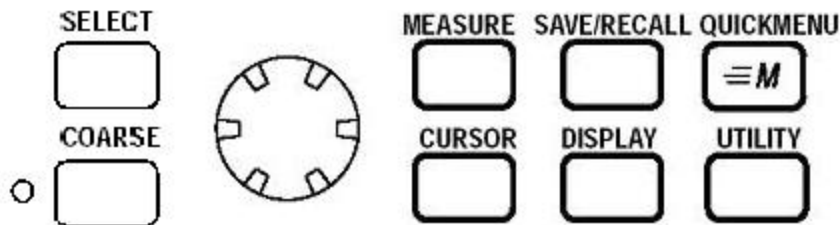
Measure, Cursor, and Save/Recall

2.5 Trigger Options

The Trigger makes signals appear to stand still on the screen rather than dancing all over the place. When the signal reaches a designated trigger level, the o-scope begins acquiring data. We will be using the Trigger MENU to adjust the triggering of the sweep of the oscilloscope display.

2.5.1 Types of Triggers

- Auto triggering:the scope will free-run in the absence of an adequate trigger signal. Useful for capturing periodic signals.

- Normal triggering: In the absence of an adequate trigger, no baseline trace will be present. Useful for capturing periodic signals.
- Single triggering: A single sweep is displayed when an acceptable trigger is detected. Useful for capturing a one-time event.
- Force triggering: Causes a trigger to occur even if there is not a signal of adequate magnitude.

TRIGGER

LEVEL

SET TO
50%

FORCE
TRIG

B TRIG
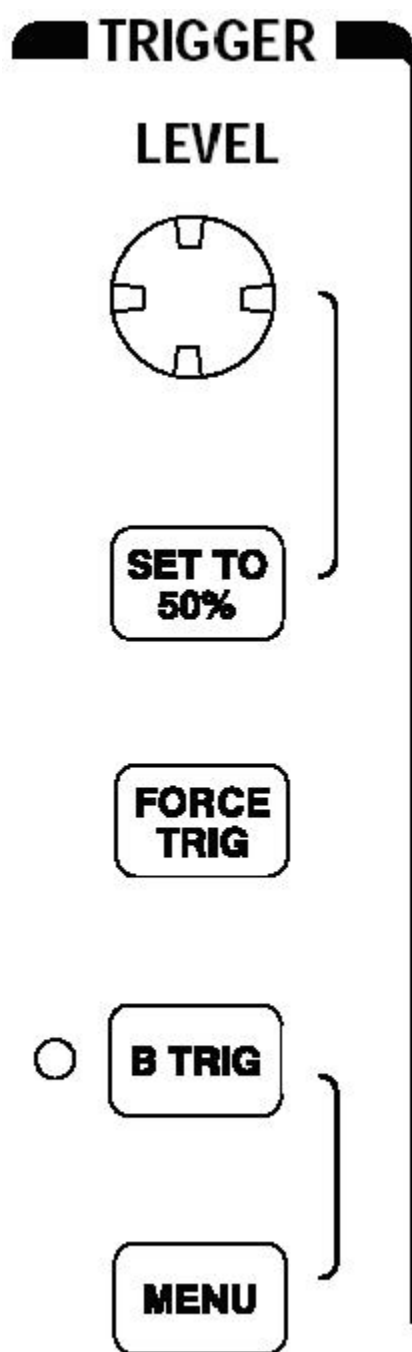
MENU

Triggering Options

2.5.2 Adjusting the Trigger

- The TRIGGER LEVEL knob selects the amplitude of the trigger level. This value is indicated by a left arrow on the right side of the display.
- Bring up the trigger menu options by selecting the TRIGGER MENU button.

1. Set the trigger mode to NORMAL.

2. Adjust the trigger level so that it is above the maximum voltage of the input signal. Notice that the trace remains fixed and that the trigger status displayed at the top right of the display goes from Trig'd to Trig?. This indicates that the channel 1 input has not met the trigger level.

3. Now switch the trigger to AUTO mode. Notice how the signal rolls when the signal is not triggered (either too high or low).

- The SLOPE option selects which slope of the signal (rising or falling) will trigger the sweep.
- The SOURCE option determines the channel that provides the trigger signal.

Practice with Trigger Options

1. Configure the function generator to produce a triangular waveform output at about 1 kHz.

2. Set the normal trigger to work on the channel connected to the function generator.

3. Make adjustments to the triggering so that you get a stable display of the triangle wave. Do not use auto triggering.

4. Set the trigger slope to rising (arrow up).

5. Adjust the amplitude of the function generator to be 1 V peak-to-peak, Vp-p,.

6. Adjust the oscilloscope sensitivity using the controls we've introduced (do not use AUTOSET) so that the waveform is as large as possible. Adjust the time scale so that at least two and no more than five complete cycles are displayed.

7. Experiment with the trigger LEVEL control. Notice how it sets the position of the waveform relative to the horizontal trigger position changes as the trigger level is adjusted.

8. You should be able to move the starting point of the trace back and forth along the ramp in the triangular wave. Again notice how the indicator switches to Trig? when the level is set above or below the top and bottom of the waveform. This indicates that the scope is not triggering and therefore not actively sampling data.

9. Set the trigger slope to Falling (arrow down) and observe that the trace now begins (relative to the horizontal trigger position) on the falling ramp of the wave.

2.6 The CURSOR button (refer to Figure 2)

CURSOR can be used to display vertical (time) or horizontal (voltage) cursors on the display.

- Cursors can be adjusted using the knob to the left of the cursor button.
- The SELECT button changes which cursor is being adjusted. The solid line is the cursor that is currently selected.
- The values corresponding to the positions of the two cursors are displayed (next to the "@" icon), as well as their difference (next to the "Δ" icon).
- Use the cursors to measure the peak voltage (Vp-p) and the period of your triangle wave.

Adjust the function generator to create a different waveform and experiment with cursors, triggers and scales.

2.7 The SAVE/RECALL button (refer to Figure 2)

Press the SAVE/RECALL button to bring up the Save/Recall menu.

- Under this menu, setup parameters and waveforms can be saved and recalled on the o-scope itself.
- The oscilloscope can save data to a floppy disk or to four internal references.
- Reference data can be analyzed like current data in the measurements and math menus.

2.8 Accessing oscilloscope data on the PC

To facilitate saving files to the hard drive of the lab computers, each of the oscilloscopes has a network interface card installed.

- Open Internet Explorer.
- Under Favorites, click the link titled o-scope. This should bring up a window that has a screen shot of the LCD display of the oscilloscope.
- Click on the tab labeled DATA.
- In this window, data from the input, math and reference channels can be saved to the computer.
- For example, using the options under Waveform transfer from the instrument, choose the Source as CH1, and the Format as SPREADSHEET.
- Click Get. A dialog box opens for the data to be saved.
- Make sure to add the correct file extension when saving the data. (.xls for Excel.)

Part 3: Data Acquisition

For some applications, it is not practical to use an oscilloscope to analyze a signal.

- Some applications generate signals that operate on timescales of hours or weeks.
- Some signals require statistical analysis
- Some data are processed by complicated algorithms.
- Some signals are used to control electromechanical systems.

For any of these applications, you would need a data acquisition (DAQ) system that measures a signal and then writes the data to a file for later analysis, or communicates with another system for real-time use.

In the next section of this lab, you will connect your signal (the function generator) to your signal conditioning equipment (the National Instruments SCXI) and data acquisition equipment (the PCI data acquisition card). You will use LabVIEW, a graphical programming language, to generate a simple Virtual Instrument (VI) to acquire data, chart the data on your monitor, and write the data to a file.

3.1 Introduction to Data Acquisition Hardware

The SCXI is used to condition a signal. The SCXIs in this lab are equipped with modules that prepare the input signal by:

- Applying a gain (magnification) to the input signal
- Filtering out high frequency characteristics
- Balancing a strain bridge

Once the signals are conditioned, they are multiplexed into a single signal. This signal is then passed by a serial cable to the data acquisition card in the PCI slot of the PC. The data acquisition card uses an analog to digital converter (ADC) to record the magnitude of an analog signal as a digital value. As a digital value, these data can be interpreted by a computer.

3.2 Configuring hardware and opening a new program

Connect the function generator output to the BNC connector on Channel 0 (the first input) of the SCXI-1520 module on the SCXI chassis.

1. Use the function generator to produce a 200 Hz sinusoidal wave.

2. Open LabVIEW 8.0.

3. Click Blank VI. Your new VI will open with two windows: the Front Panel (the user interface) and the Block Diagram (which is where you develop your graphical code). These two windows are linked, and as you develop your VI, you will see that some objects that you place in one of the windows will appear simultaneously in the other window.

4. On either the front panel or the block diagram, click File>>Save As. Save your VI to the folder (on the desktop) that corresponds to your lab section. (Saving either window will save the content from both the front panel and block diagram windows.)

3.3 Acquiring data

You will now develop your VI to acquire the data that the ADC is sampling. The VI will acquire 1000 samples at a rate of 100,000 samples per second. Once the 1000 samples have been gathered, the data will be analyzed programmatically to determine the peak-to-peak amplitude and the DC offset. The data set will be displayed on a graph, and will be written to a measurement file that can be opened in Excel.

LabVIEW contains a useful set of functions for data acquisition known as DAQmx. DAQmx is NI's driver configuration software for DAQ devices. We will use the DAQ Assistant function to open communication with the DAQ hardware, read data from the device, and close communication.

5. On the block diagram, right click to bring up the Functions Palette. (Refer to Figure 4.) The Functions Palette contains all of the functions and structures that can be placed directly on the block diagram. Notice that the functions are organized into sub-palettes by functionality.
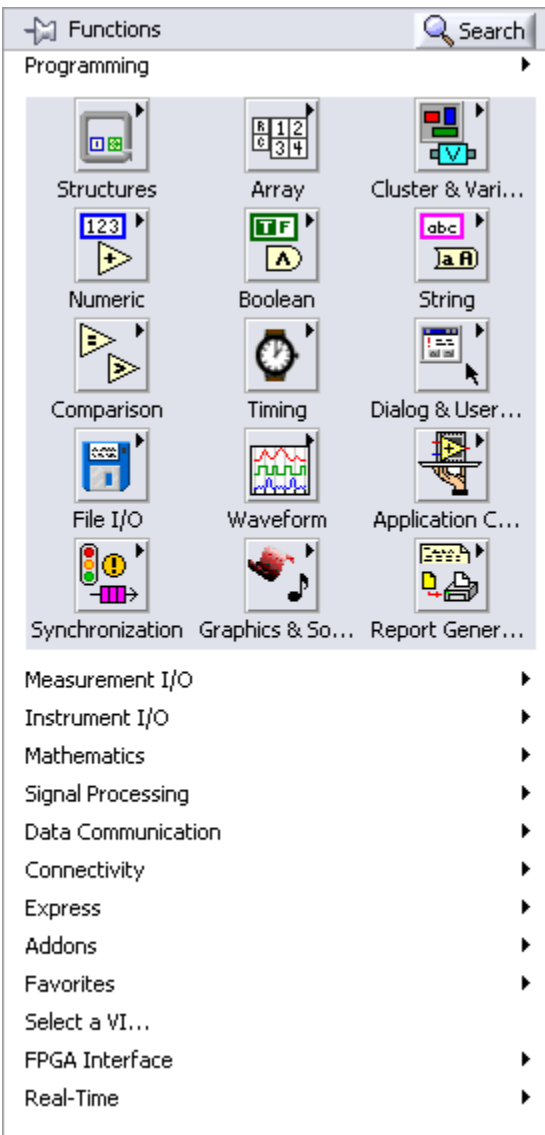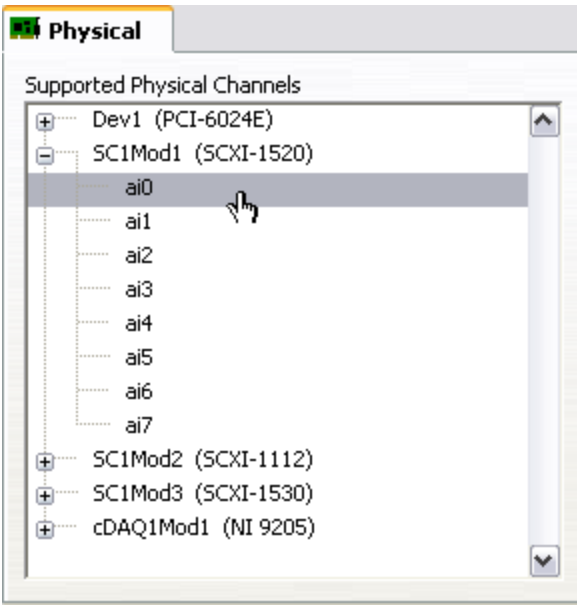
Functions Palette

6. Click Measurement I/O>>NI-DAQmx>>DAQ Assist.

7. When you place the DAQ Assistant on the block diagram, a Create New… window will open as shown in Figure 5. This menu will guide you as you configure your software to communicate with your hardware.
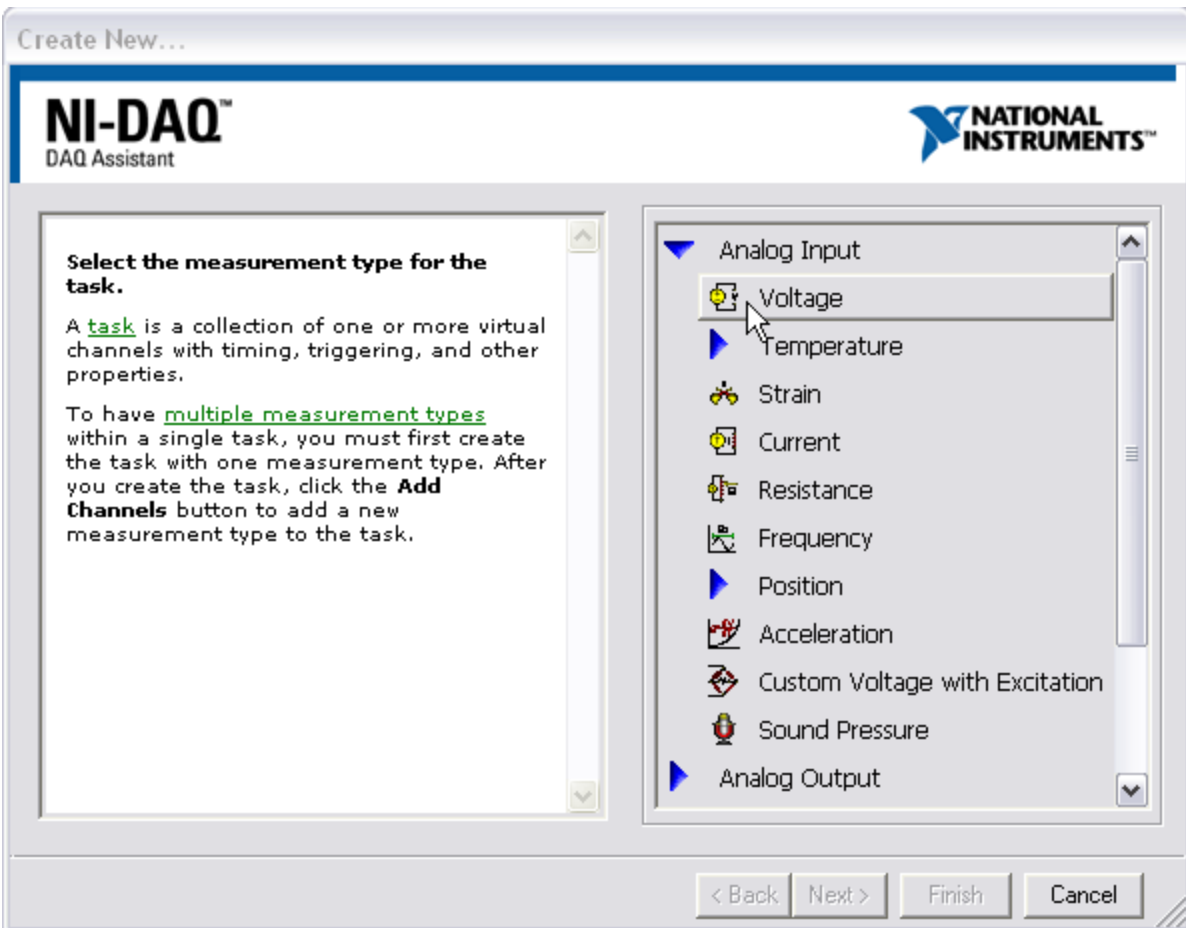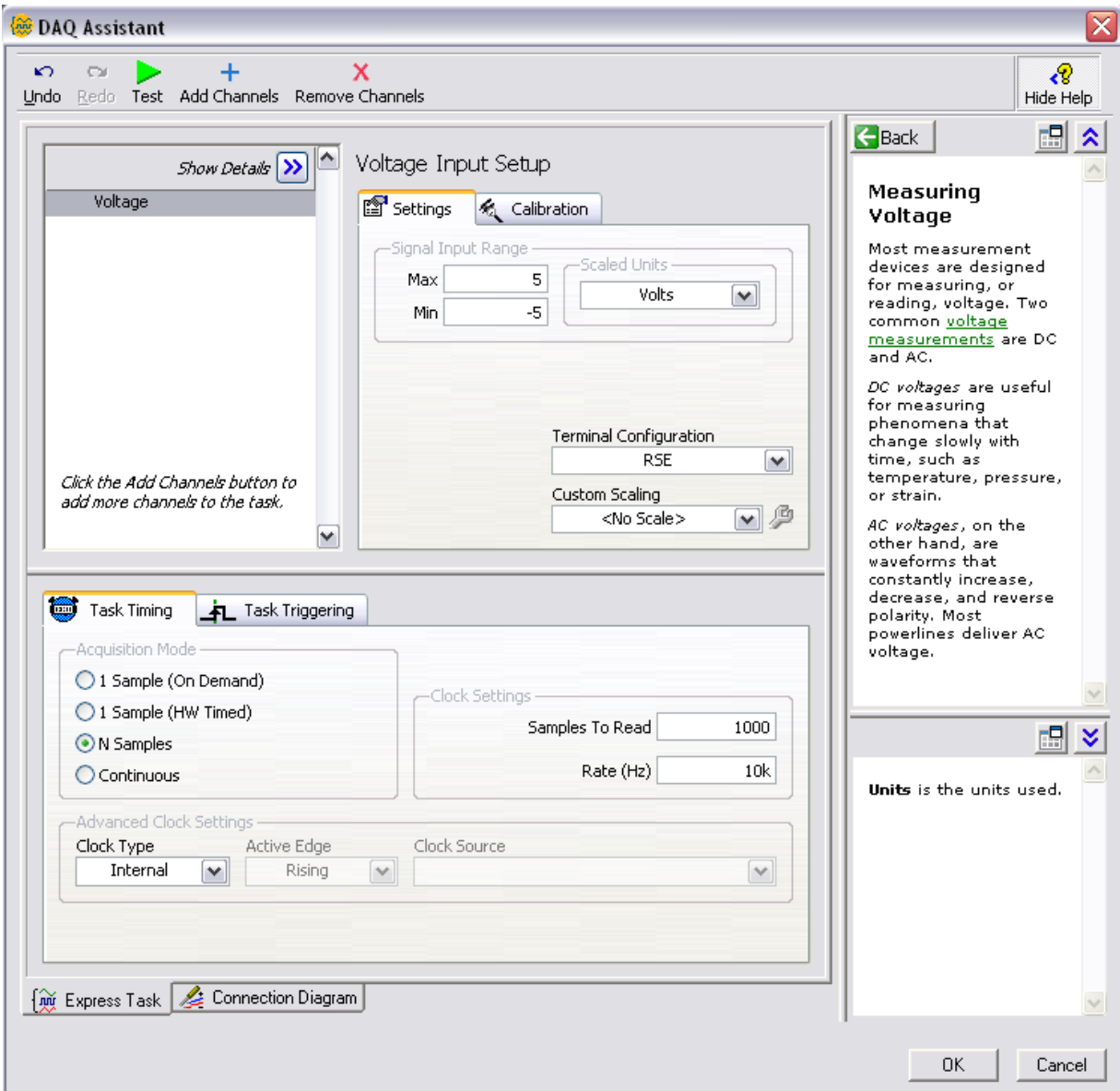
Create New DAQ Assistant Window

8. Select Analog Input>>Voltage. DAQmx will scan the system for available hardware and present the options; as shown in Figure 6.

Physical

Supported Physical Channels

- Dev1 (PCI-6024E)
- SC1Mod1 (SCXI-1520)
  - ai0
  - ai1
  - ai2
  - ai3
  - ai4
  - ai5
  - ai6
  - ai7
- SC1Mod2 (SCXI-1112)
- SC1Mod3 (SCXI-1530)
- cDAQ1Mod1 (NI 9205)

List of Physical Channels

9. Expand the cDAQ module that is labeled (SCXI-1520).

10. Select ai0, (the analog input channel that the function generator is connected to.)

11. Click Finish.

DAQ Assistant Configuration Window

12. The DAQ Assistant window should now open. Configure the DAQ Assistant as shown in Figure 7.

- Set the input range from 5 to -5 volts.
- Change the Terminal Configuration to Differential. (Differential assumes that the voltage is measured between two channels, rather

than referenced to a common ground.)
- Set the Acquisition Mode to N Samples.
- Change Samples to Read to 1000 and the Rate to 100,000.

13. When you click OK, LabVIEW will build a subVI according to your specifications.

3.4 Amplitude Measurement of Data

Similar to the measurements made on the oscilloscope, LabVIEW programs can analyze characteristics of a signal.

14. Right click on the Block Diagram to bring up the Functions Palette. (Note that if you click on the Front Panel you will bring up the Controls Palette, which has entirely different choices.)

15. Select Express>>Signal Analysis>>Tone. The Configure Tone Measurements dialog box will now open. The Tone Express VI will display values related to the amplitude and frequency of the signal.
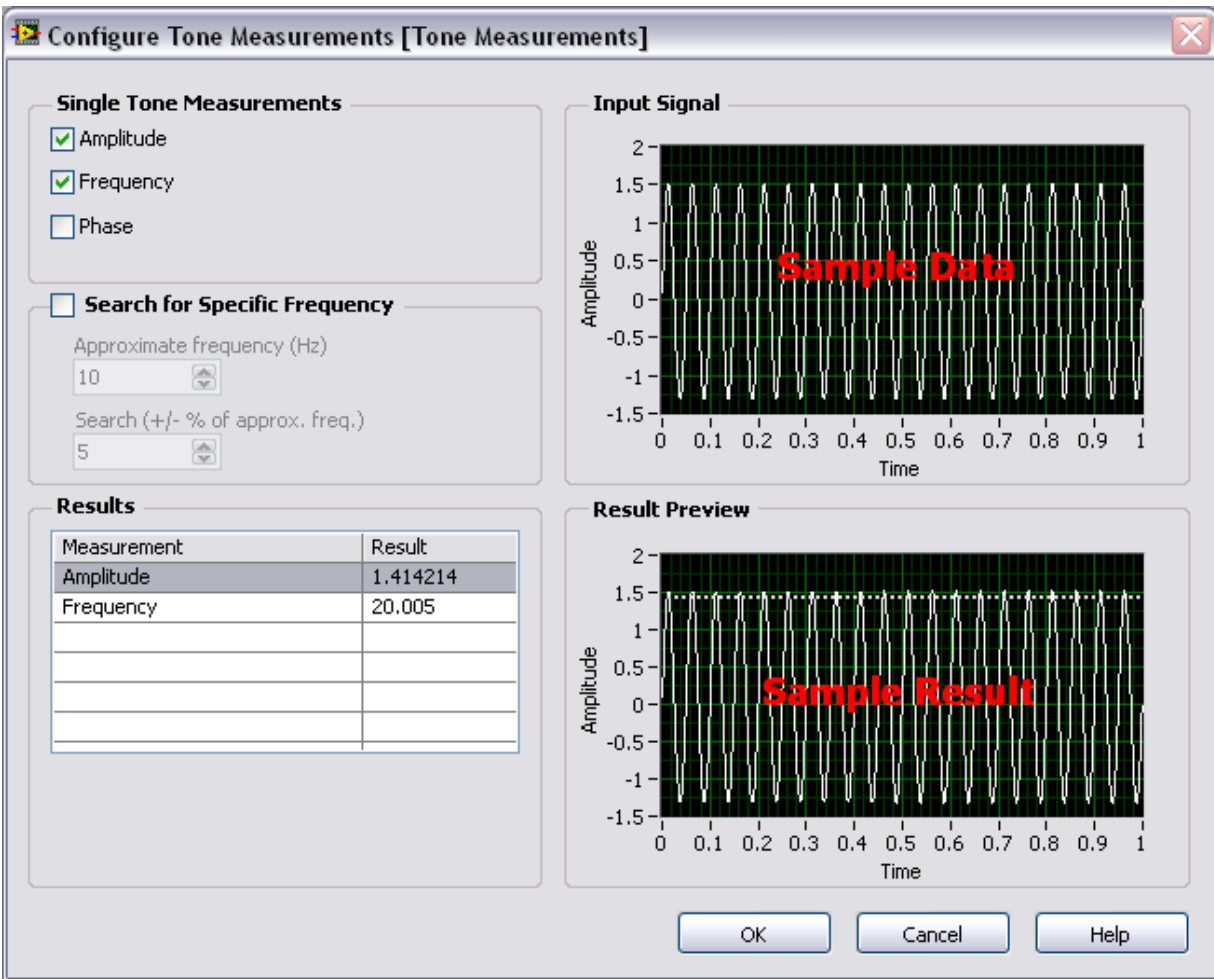
Configure Tone Measurements Box

16. Configure the dialog box as shown in Figure 8.

17. Select Amplitude and Frequency in the Single Tone Measurements box.

18. When you click OK, LabVIEW will build a subVI according to your specifications.

19. Place your cursor over the data output terminal on the DAQ Assistant icon. The cursor should change to a wire spool. Click on the data terminal and connect the wire to the signals inputterminal on the Tone Measurements

icon. This wire represents the data flow from the DAQ Assistant to the measurements subVI.

20. For both output terminals on the Tone Measurements icon, right click and select Create>>Numeric Indicator to create Front Panel displays of these parameters.

21. Your block diagram should now be similar to Figure 9.

22. There are several Express VIs in the Signal Analysis palette that are similar to the Tone Express VI. These other Express VIs display information regarding frequency content, DC offset, curve fitting, etc. Depending on time, you may want to experiment with these other functions.
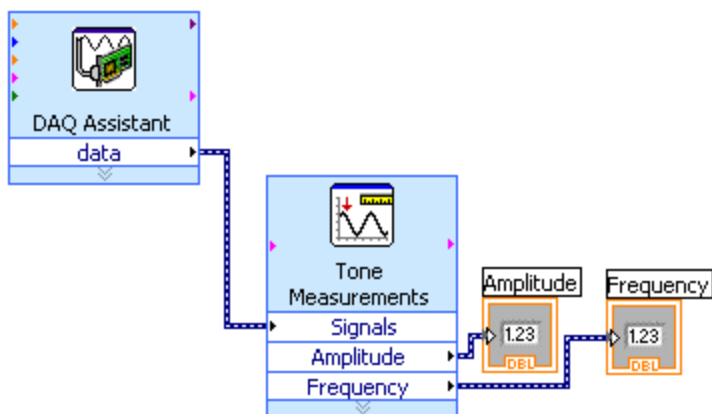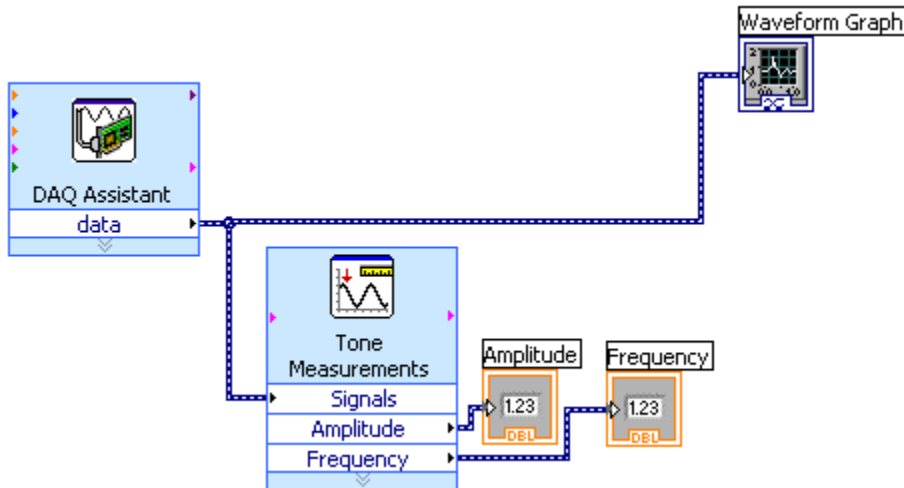

Block Diagram with Measurements
SubVI

3.4 Displaying data on-screen

One of the strengths of LabVIEW is the ease with which user interfaces are developed. In this section we will develop an on-screen graph display.

23. Right-click on the wire between the DAQ Assistant and the Tone Measurements icons. Select Create>>Graph Indicator.

24. Display the Front Panel by pressing <Ctrl-E>. You should see that a graph has been created on the Front Panel.


Block Diagram with Graph Added

3.5 Writing Data to a File

Now that the VI can acquire data, we will develop a means to write the data to an Excel file and display the data onscreen. We will use the "Write to Measurement File" Express VI.

25. Right click on the Block Diagram to bring up the Functions Palette.

26. If you can't find the "Write to Measurement File" Express VI, click on the search button at the top of the Functions Palette. Type "write to file" and double click the choice that ends in <<file I/O>>. The search menu will change into the Functions Palette, and the "Write to Measurement File" Express VI will be highlighted. Place this Express VI on the block diagram.
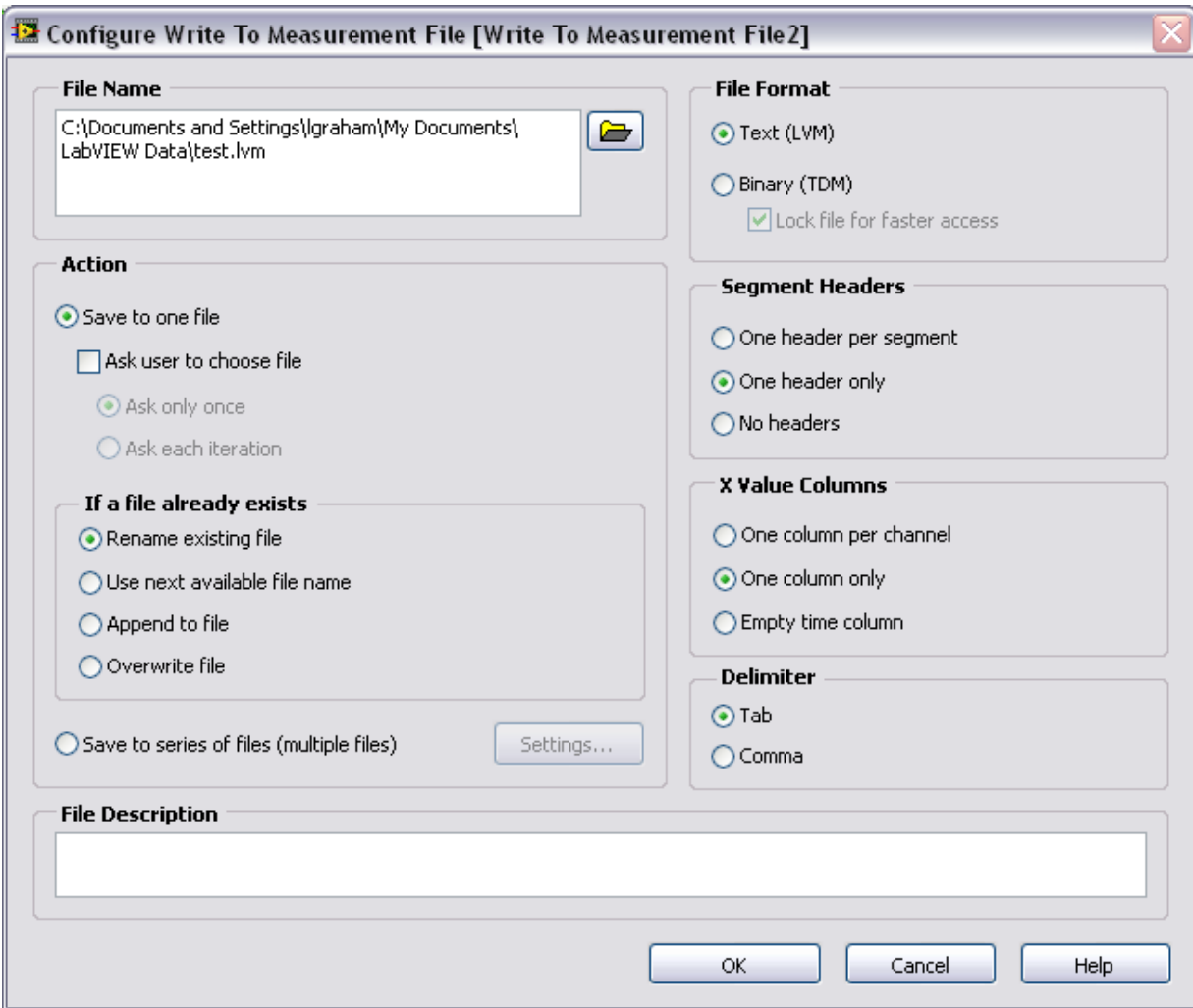
Write to Measurement File Configuration Window

27. Configure the window as shown in Figure 11. Configure the File Name window to contain the path to wherever you wish to store your data. Change the Segment Headers to One Header Only and the X Value Columns to One Column Only. Click OK.

28. Place your cursor over the data output terminal on the DAQ assistant icon. The cursor should change to a wire spool. Click on the data terminal and connect the wire to the signals inputterminal on the Write to File icon.

29. Right click the File Name terminal input on the Write to File icon. Select Create>>Control. This creates a field on the Front Panel where the user can enter the name of the data file that will be created.

30. Create a write disable button by right-clicking on the front panel. Click Boolean>>Push Button. Type "Store Data" to change the name of the control.

31. On the block diagram, connect the push button to the Enable terminal on the Write to File icon. (When this control is enabled, the program will write data to the file-path indicated on the Front Panel.)

32. Your Block Diagram should now be similar to Figure 12.
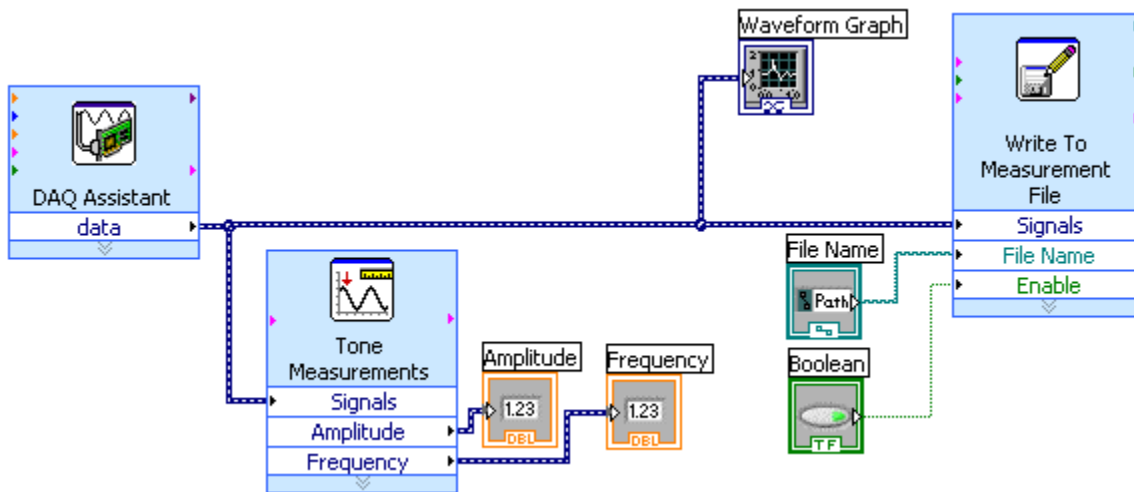
Block Diagram

3.6 Running the VI

33. On the Front Panel, click the Run button. You should see the signal from the function generator displayed on the graph.

34. When you enable data storage, a .lvm file should will be created in your folder. On the Windows desktop, open your group's folder. Right click on the .lvm file that you just made and select Open With>>Microsoft Office Excel.

35. Plot the data and compare it to the graph shown on the front panel. You may want save screen shots or make hard-copies of any plots to include in your lab book.

Part 4: Shutting Down

After you have saved any data you want to keep, close all programs and log off of the computer. Turn off the CompactDAQ chassis, the function generator, and the oscilloscope. Take time to clean up your work area. Ten percent of your lab book grade will be based on cleanliness.

Lab Report

You are to submit a one-page memo report describing what you learned from the lab. See the Lab 1 Writing Assignment handout for more details.

Temperature Measurement and First-Order Dynamic Response

Introduction:

In this lab exercise, you will determine how various temperature measurement systems respond to different inputs. This analysis will include a thermometer, a thermocouple, and a thermistor.

Teaching Objectives:

The purpose of this laboratory is for each student to understand and do the following:

1. Instrument calibration.

2. Measurements with thermocouples and thermistors.

3. Determination of time constants and comparisons of the dynamic responses of three measuring systems.

4. Application of first-order dynamic behavior to dynamic temperature measurement problems.

Preparatory Reading:

Figliola and Beasley

Measurement System Behavior, First Order Systems: pp. 72–87

Thermistors: p. 295

Thermoelectric Temperature Measurement: pp. 300-313

Overview:

In this lab, we will use first-order models to approximate the response of a thermometer, a thermocouple, and a thermistor to temperature inputs. Each

of these temperature sensors measure temperature in a different way.

- A thermometer senses a change in temperature as a change in the density of a fluid.
- A thermocouple consists of two wires of different metals joined at one end (the junction). When a voltage is applied across the free ends of the two wires, the differing properties of the wires create an induced voltage that it proportional to the temperature change at the junction.
- A thermistor is a thermally sensitive resistor. When the temperature of the thermistor changes, a proportional change in resistance occurs.

Procedure:

Part 1: Modify a VI for temperature measurements

In this lab, we will build on the skills that were developed in Introduction to Benchtop Equipment and Data Acquisition. It is assumed that the student understands the concepts of a Front Panel, a Block Diagram, the Control Palette, and the Functions Palette. It is also assumed that the student can search the control and functions palette for necessary features.

1. Connect a thermocouple to channel 0 of the SCXI-1112 module.

2. Open LabVIEW 8.0.

3. Open the VI that you developed for Introduction to Benchtop Equipment and Data Acquisition.

4. Click File>>Save As…>>Substitute Copy for Original to save your VI with a new name.

5. On the Block Diagram, double click the DAQ Assistant to open the configuration dialogue box.
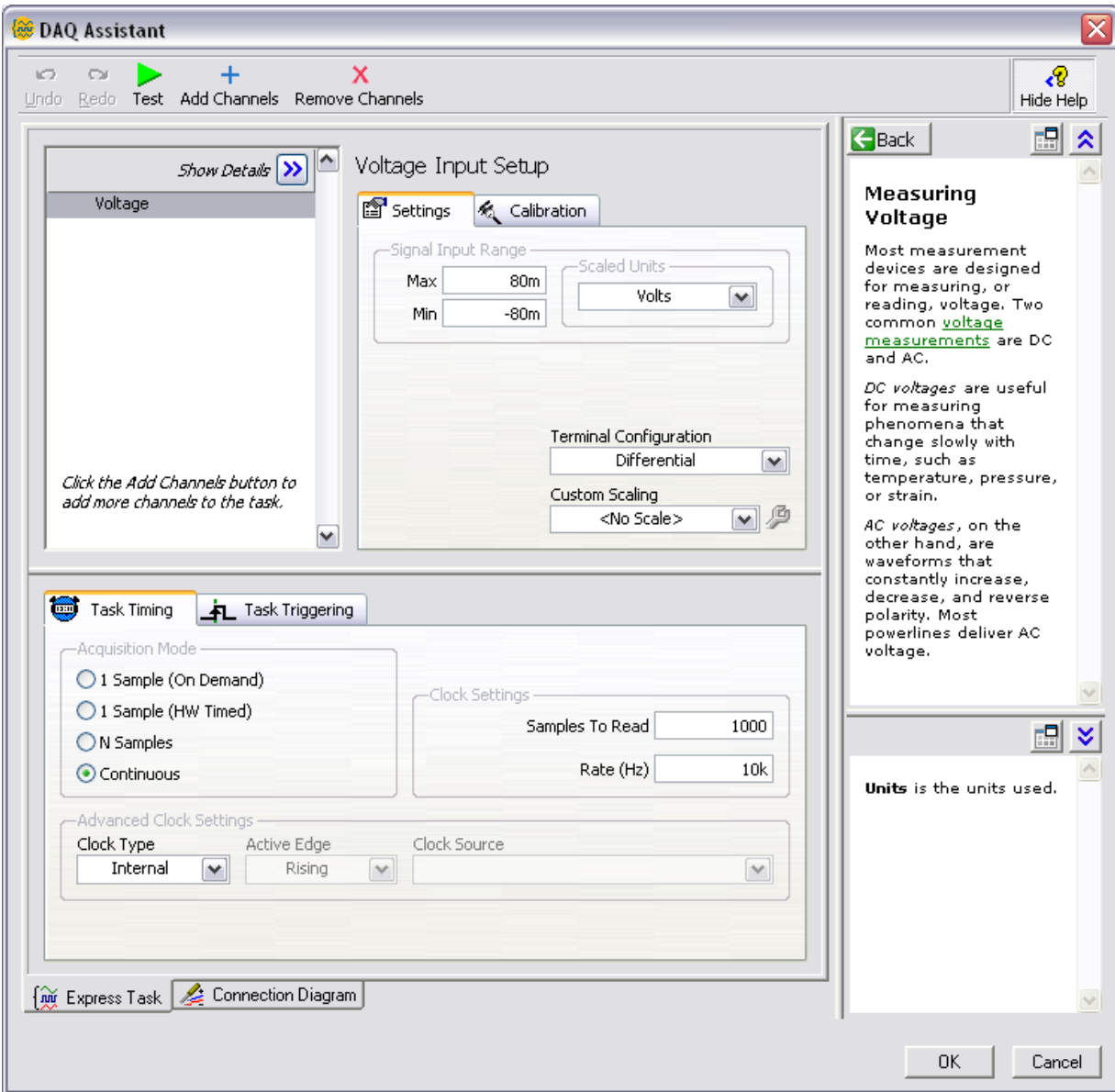
DAQ Assistant Configuration Box

6. Configure the DAQ Assistant as shown above.

1. 1. Click Voltage and press Delete to delete the channel you used in
   the first lab.
   2. A dialog box will ask if you want to delete the selected channel.
   Click Yes.

3. Add a new measurement channel.

   1. Click Add Channels>>Voltage.
   2. Select the channel that the thermocouple is connected to (SC1Mod2>>ai0).
   3. Click OK.

   4. The input range should be 80mV max to -80mV min. (Thermocouples have a low voltage output.)
   5. The Terminal Configuration should be Differential. (The ground terminal is not used.)
   6. Change Acquisition Mode to Continuous.
   7. Click OK.
   8. A dialog box will open asking if you would like a While Loop to be made around the DAQ Assistant. Click No. (You will place a While Loop in the next step that includes the subVIs that are already on the Block Diagram.

7. Place a While Loop around the entire Block Diagram as shown below. LabVIEW While Loops are similar to While Loops in any text-based programming language. The routine inside the loop will run repeatedly until a true value is passed to the loop condition, represented by a red button at the bottom right corner of the loop.
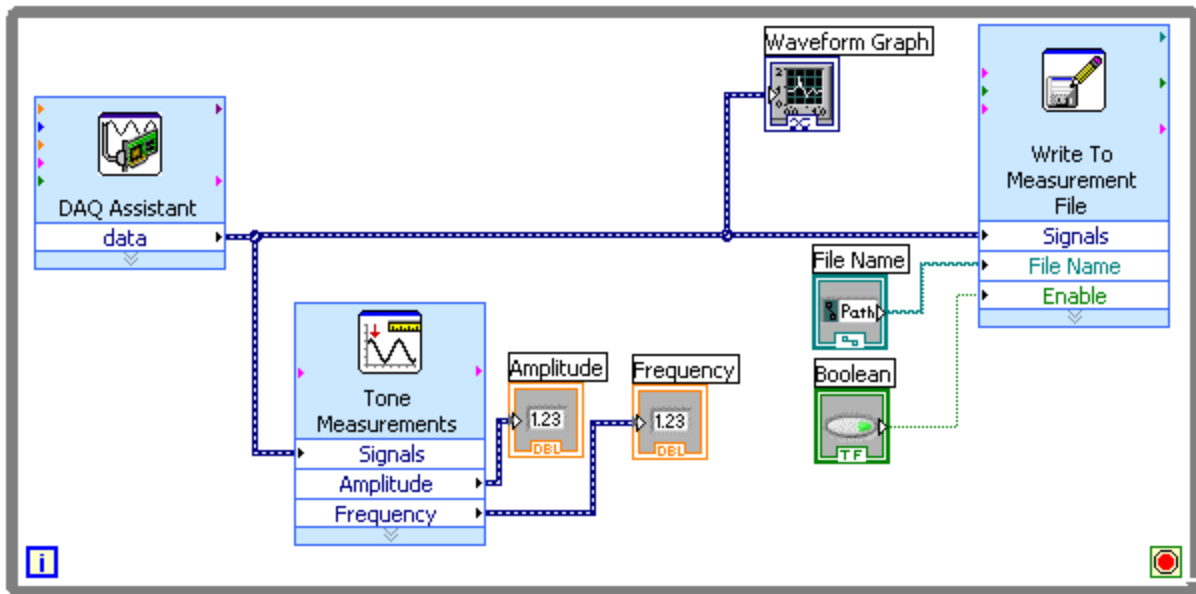
Block diagram with while loop

8. A toggle button can be used to end the While Loop routine. Right click on the loop condition icon and click Create>>Control.

9. Double click the Write to Measurement File ExpressVI to open the configuration box. Under If a file already exists, select Append to File. Click OK. (Because the Write to File function now occurs in a loop, the data must be appended to the end of the file, rather than overwriting the data from the previous loop.)

10. On the front panel, place a Thermometer.

11. Right Click the Thermometer. Click Visible Items>>Digital Display.

12. On the Block Diagram, inside the While Loop, place a Time Delay Express VI. When the Configuration Box opens, change the Time Delay (seconds) to 0.001. (Without a delay in the While Loop, the program will repeat as quickly as it can and consume processor power.)

13. Your Block Diagram should resemble Figure 3.

14. Click File>>Save As…>>Copy>>Create Unopened Disk Copy. Save the VI with "Lab3" in the file name. (You will use the VI as it is now as a starting point in Operational Amplifiers and Signal Conditioning.)

15. Save a second copy with "Lab4" in the file name. (You will also use the VI as it is now as a starting point in Strain Gages and Force Measurement.)
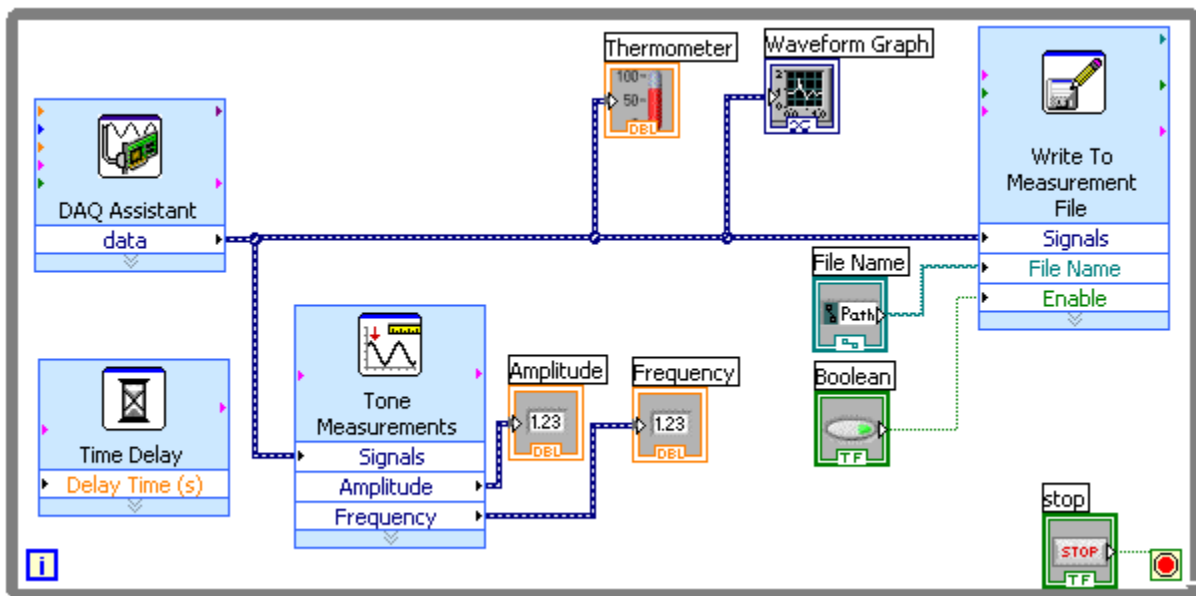
Completed block diagram

Part 2: Calibration Curve

You will create a static calibration curve to correlate voltage measurements with temperature readings. We will use the thermometer as the standard. (Note: We use the thermometer as a standard for convenience. Do not assume that an alcohol-in-glass thermometer is a temperature measurement device with superior accuracy.)

2.1 Calibrate the thermocouple

1. Take temperature readings with the thermometer in ice water and boiling water. Be careful not to allow the devices to come into contact with the hot plate or the bottom of the beaker.

2. Record the temperatures measured.

3. Use the thermocouple to measure both temperatures. Again, be careful not to allow the devices to come in contact with the hot plate or the bottom of the beaker.

4. In Excel:

   1. 1. Plot the temperatures measured with the known standard values on the y-axis and the ADC output (voltage) resulting from the thermocouple input on the x-axis.
       2. Create a linear curve fit using the two data points.

The linear fit results in an equation that relates the ADC output obtained from the thermocouple to the known temperature from the thermometer:

$T = k(Vadc) + b$

where "T" is the temperature, "k" is the gain (°C/V), "Vadc" is the voltage read in to the computer, and "b" is the zero-voltage offset. The gain "k" is the inverse of the static sensitivity, which has units of V/°C.

5. Repeat the calibration procedure for the thermistor.

2.2 Modify your VI for calibration
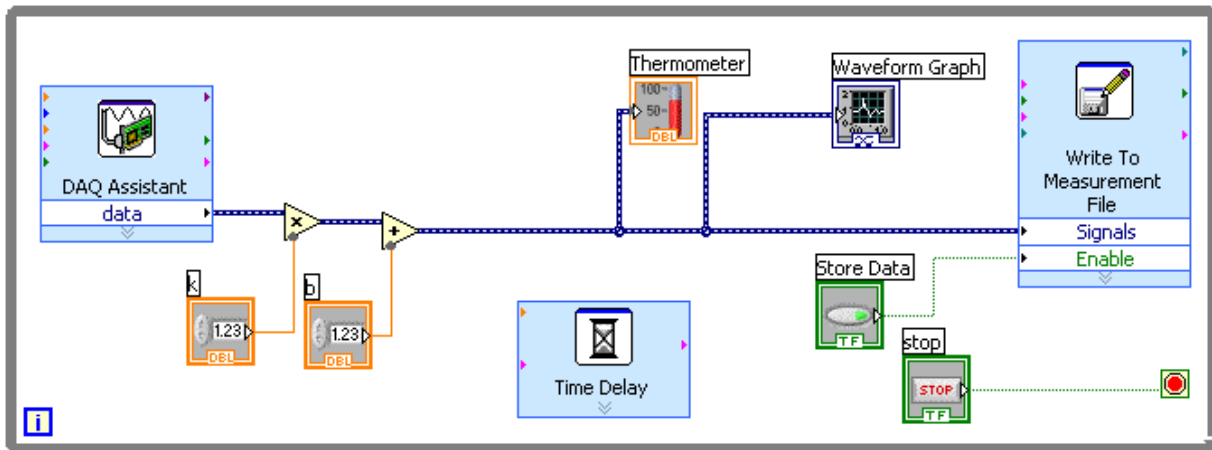
Modified block diagram

In this section, you will multiply the data by the conversion scalar "k" that you found experimentally. You will also add an offset "b".

1. Delete the Tone Measurement ExpressVI. Delete the Frequency and Amplitude indicators.

2. Delete the wires that connect the DAQ Assistant to the thermometer, the Waveform Graph, and the Write to File ExpressVI. Press Ctrl+b to remove any broken wire remnants.

3. Place a Multiply and an Add on the block diagram.

4. Create a control and connect it to the y terminal of the Multiply icon. Rename the control "k".

5. Create a control and connect it to the y terminal of the Add icon. Rename the control "b".

6. Wire your block diagram as shown in Figure 4.

7. On the front panel, enter the values you found experimentally for the "k" and "b" values.

8. When you run the VI, your measurements should now be displayed as temperatures.

Part 3: Time Constants

The calibration curves you found represent static calibrations. In the following section, you will consider the dynamic behavior of the sensors. The dynamics of all the temperature sensors used in this lab can be modeled accurately as first-order systems. The natural response of a first-order system can be characterized by a single parameter — the time constant.

Time constants can be found by observing the response of the instrument to a step input in temperature. Note that the second medium governs the physics of the transient response. For each of the sensors, you will determine the time constant for transfers from ambient air to ice water and from hot air to ambient air.

An ice water bath will be provided for your tests. A heat gun will be used to generate hot air. Caution: Keep all temperature sensors (particularly the thermometer and the thermistor) at least 9 inches away from the heat gun. Holding them closer to the heat gun will result in temperatures outside the operating range of the sensors.

Four methods will be used to calculate various time constants. Table 1 summarizes the tests to be performed. The testing methods are described below.

Table 1. Time constant calculation methods.

	thermometer	thermocouple	thermistor

ambient air to ice water	1, 2	3, 4	3, 5
hot air to ambient air	1	3	3
ice water to ambient air	-	3	-

Method 1- Time Elapsed at 63.2% Sensor Accuracy

Thermometer: Perform this test for ambient air to ice water. Repeat for hot air to ambient air.

1. Record the temperature of the initial medium, T0, and the temperature of the final medium, T_inf.

2. Calculate the intermediate reference temperature, Tr that is 63.2 percent of the way from T0 to T_inf.

3. With the instrument at steady state (reading T0) in the initial medium, move it quickly to the final medium.

4. The time from when the step in temperature began to when the instrument reads Tr is the time constant, . This time can be measured using a stopwatch.

Method 2 - Mathematic Definition of Tau

Thermometer: Perform this test moving from ambient air to ice water.

1. Record the temperature of the initial medium, T0, and the temperature of the final medium, T_inf.

2. Choose an arbitrary reference temperature, Tr that is between 60 and 80 percent of the way from T0 to T_inf.

3. With the instrument at steady state (reading T0) in the initial medium, move it quickly to the final medium.

4. Record the time trthat it takes to reach the reference temperature Tr.

5. The following expression can be used to calculate the time constant tau:

$$e^{-(t_r/\tau)} = \frac{T_\infty - T_r}{T_\infty - T_0}$$

    equation (1)

Method 3 – Analyze Acquired Data

Thermocouple: Perform this test for ambient air-to-ice water, hot air-to-ambient air, and ice water-to-ambient air.

Thermistor: Perform this test for ambient air-to-ice water, and hot air- to-ambient air.

1. Run the VI.

2. Insert the thermocouple into the initial medium.

3. Click the Enable toggle control as you transfer the thermocouple to the final medium.

4. When the sensor comes to equilibrium, stop the VI.

5. Open your measurement file in Excel.

6. Determine the time required for the sensor to complete 63.2 percent of the transient from the initial temperature to the final temperature.

7. Change the file path to prevent overwriting data. (You can either change the name of the file after it is written, or you can change the name of the path that the next file will be written to.) You will use the saved data for subsequent calculations.

Method 4- Error Fraction

Thermocouple: Complete this step using the data from the ambient air-to-ice water transfer.

Thermistor: Repeat this step using the data from the ambient air-to-ice water transfer.

Equation 2 below gives an expression for the error fraction.

$$\Gamma(t) = \frac{T_\infty - T(t)}{T_\infty - T_0}$$

   equation
    (2)

1. Calculate the error fraction at each instant of time for the data you have gathered.

2. Take the natural log of the error fraction data.

3. Plot the natural log of the error fraction versus time.

4. Find the negative reciprocal of the slope of this curve. This is the time constant.

The error fraction versus time curve provides an easy way to judge the accuracy of a first-order model for the data gathered. If the error fraction plot is linear, then the first-order model is a good approximation of the dynamic behavior of the system. (See pages 78–79 of Figliola and Beasley for a more detailed discussion of the error fraction and its use to calculate the time constant.) Are the thermocouple and thermistor in water adequately modeled by the first-order system approximation?

Method 5- Least Squared Error

Thermistor: Complete this step using the ambient air-to-ice water data.

Equation (3) below gives the theoretical expression for the temperature, T(t), as a function of time when the temperature of the environment is suddenly changed from T0 to T∞. The thermocouple is assumed to be at temperature T0 when the change occurs at t = 0.

$$T(t) = T_\infty + (T_0 - T_\infty)e^{-t/\tau}$$

    equation (3)

1. Using Excel, compute a value for tau so that the square of the error between the theory and your experimental result is minimized. This is called a least square error approach and is very common in engineering.

2. Plot both the test data and the results from Equation (3) on the same axes to compare how well this model fits the data.

Part 4: Additional Tests

Once you have performed all of the tests outlined above, consider the following tests:

•Transfer the thermocouple from ice water to room temperature air.

How does the time constant compare with the one obtained from the air-to-ice-water transition? Is it different? Why or why not?

•When calculating the time constant for the thermocouple in transition from air to ice water, you measured the time to go 63.2 percent of the way from the initial temperature (around 21 C) to the final temperature (around 0 C). Using the same graph, recalculate the time constant using14 C as the starting temperature (i.e., what is the time required to go 63.2 percent of the way from 14 C to 0 C?). Is this time constant value significantly different from the initial value you computed? Why or why not?

Part 5: Thought Questions

Document your response to these questions in your lab book.

1. How do the time constants of the thermometer, thermocouple, and thermistor compare? Why is the thermometer "slow" compared to the other devices?

2. How do the time constants obtained for air and water compare? Are they significantly different? If so, explain why.

3. Does the instrument need to be at steady state before transferring it to the final medium?

4. How accurate (qualitatively) is the first-order model approximation for the different sensors and media?

Part 6: Frequency Response

From the measured time responses of the thermometer, the thermocouple, and the thermistor, it can be seen that the response time of these sensors is not instantaneous. From this we can infer that these measurement systems may have some difficulty following inputs that change quickly. Frequency response is a means of characterizing the dynamic response capabilities of a system.

The frequency response of a system can be determined by calculating the response of a system to sinusoidal inputs at different frequencies. Two important characteristics are:

- the magnitude ratio between the output and input waveforms
- the phase difference between the output and input waveforms

For a first-order system, we can create the frequency response plot using only our knowledge of the system time constant. The magnitude ratio can be calculated from the following expression.

$$M(\omega) = \frac{1}{\sqrt{1+(\omega\tau)^2}}$$

equation
(4)

The phase difference is determined from expression 5.

$$\phi(\omega) = -\tan^{-1}(\omega\tau)$$

equation
(5)

In Excel, create a frequency response plot for the thermometer, thermocouple, and thermistor using the time constant from the ice water tests. Plot the magnitude ratio on a log-log scale and the phase difference on a semilog scale (log scale of frequency axis). Consider frequencies from 0.1 rad/s to 1000 rad/s. Your plots will look nice if you use ten points per decade of frequency (i.e., 0.1, 0.2, 0.3, … ,1, 2, 3, … , 10, 20, 30, …). Overlay the three magnitude responses on one plot and the three phase responses on another. How do the frequency response characteristics of the thermometer, thermocouple, and thermistor compare?

Part 7: Dynamic Temperature Measurement Problem

Include the analysis for this problem in your lab book.

Assume that the thermocouple you used is placed in a water stream at a point where the temperature varies sinusoidally between 60 C and 80 C at a frequency of 40 rad/s. Assuming that the thermocouple time constant is the same as the value you determined in the ice water case, what will be the range of sinusoidal variation in the temperature readings? Repeat the calculations for a temperature frequency of 2 rad/s. Now assume the thermocouple is in air instead of water. Determine the range of the variation using input frequencies of 10 rad/s and 1 rad/s.

Clean Up:

Please straighten up your lab work area. Turn off the data acquisition hardware and leave your work area clean. Your help in keeping the lab neat is appreciated.

Report:

For this lab, you are to write up the Method section of a full formal report. Guidelines are given in the example report in the Undergraduate Guide.

Avoid presenting results in the Methods section. Your writing assignment is relatively short. Take the time to do a good job. Since you will not be submitting Results or Discussion of Results in the report, you will need to document your results in your lab book. Also, be sure to answer the questions from the lab handout in your lab book.

Operational Amplifiers and Signal Conditioning

Introduction:

In this lab, you will build several basic op-amp circuits to learn about the signal conditioning of sensor signals.

Teaching Objectives

- Understand how an op-amp works.
- Learn about different types of op-amp circuits and their uses.
- Gain practical experience in implementing op-amp circuits.
- Build a voltage divider, an inverter, a differential amplifier, a gain circuit, and low-pass filter circuits.
- Demonstrate the filtering capability of the low-pass filter and its use to prevent aliasing of sampled data.

Preliminary Reading

Figliola and Beasley

Analog Signal Conditioning: 211-226 (Sections 6.6-6.8)

Component Identification

In this lab, we will be using several types of electronic components: resistors, capacitors, trim potentiometers, and op-amps.
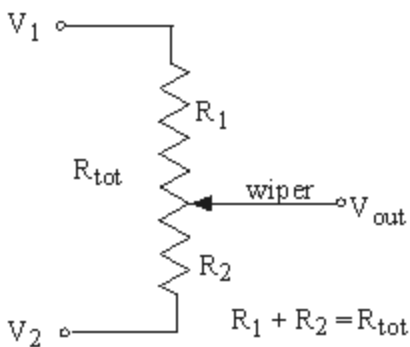
Resistors

Resistors come in a wide range of resistances. For op-amp circuits, values in the range of 10 k ohm to 250 k ohm work best. For this lab, we will use primarily 10 k ohm and 100 k ohm resistors and maybe a few others.

Resistors are color coded so that with some practice you can decipher resistance values from the color bands on the resistors you are using.

Capacitors

Capacitors also come in a wide range of values and are made using a variety of techniques. The capacitors that we will use for signal conditioning are fairly small and have capacitances measured in Pico farads or microfarads. Some capacitors are polarized and care must be taken to ensure that they are connected to a circuit with the proper orientation.

Trim Potentiometer
Schematic

Trim Potentiometers

A trim potentiometer is a variable resistor. It has three leads.

1. Positive power supply
2. Wiper
3. Reference power

The resistance of the trim pot is measured between the wiper and one of the other two leads. A trim-pot is depicted schematically in Figure 1. An adjustment screw moves the wiper along the length of a coil resistor. The resistance between lead 1 and the wiper increases or decreases as the wiper moves further or closer.

The output voltage (Vout) is related to the input voltages by the following relationship:

$$V_{out} = \frac{R_2}{R_1 + R_2} V_1 + \frac{R_1}{R_1 + R_2} V_2$$

   equation (1)

**Operational Amplifier (Op-Amp)**

For this lab, we will use the LM741 op-amp. The LM741 is a very common op-amp package, with one op-amp per integrated circuit (IC). (Many op-amp packages have two or four op-amps per IC.) A schematic representation of the LM 741 is shown in Figure 3. The semi-circular depression on the chip marks the top of the chip and the pins are numbered counterclockwise around the chip. The highest number pin is always opposite pin 1.

LM741 Pinout:

1. Null offset adjustment (not used)
2. Inverting (-) input
3. Non-inverting (+) input
4. Connect to -12 V
5. Null offset adjustment (not used)
6. Output

7. Connect to +12 V

8. Not used

Data sheets for this and other integrated circuits (IC's) can be obtained from the National Semiconductor web site (www.national.com).
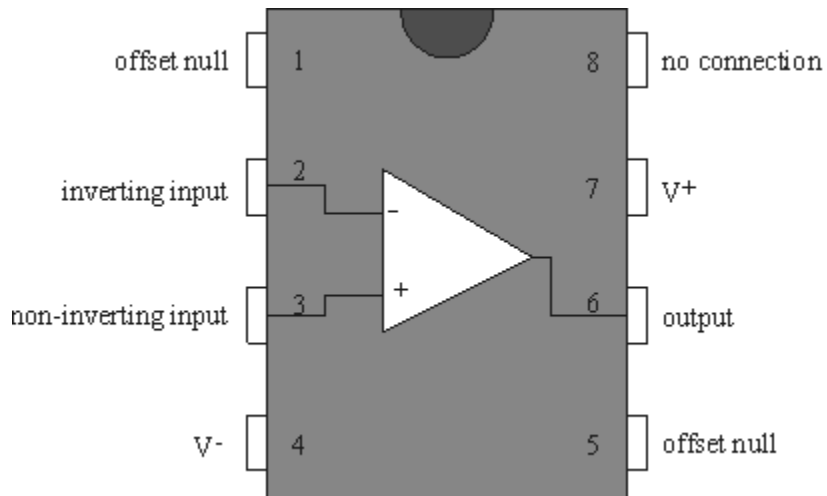
LM741 Connection Diagram

The Prototyping Board

You will build circuits for this lab using a solderless prototyping board known as a proto-board or breadboard. Breadboards allow us to connect circuits by plugging in components and wires. No soldering is required. A drawing of a breadboard is shown in Figure 4 below. As indicated by the gray lines, horizontal rows of connectors in areas B and C are internally connected. When a chip is plugged into the board as shown, these strips give you four connections to each of the IC pins. The connections in regions A and D are intended as power and ground buses. As indicated by the gray lines, the columns in these regions are tied together vertically.
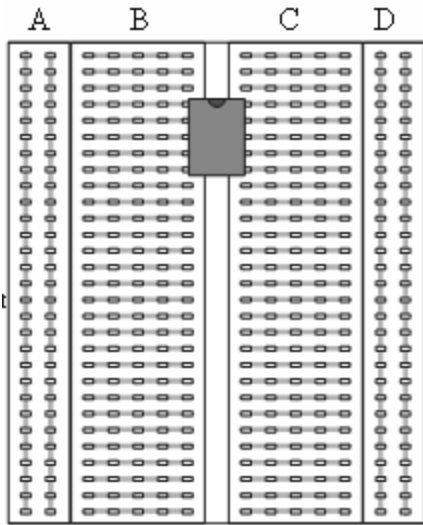
Breadboard

The breadboards are fairly robust; however they are susceptible to damage from wires that are too large. Do not use wires larger than 28 gauge. Components with leads that are too large can damage the breadboard as well.

Procedure

Part 1: Set Up the Power Supply

You will use a Tektronix triple-output power supply. The power supplies should be set up when you come to lab. For your reference, set-up instructions are given below. Perform the following steps before you turn the power supply on.

The power supply has three dc-power outputs; one capable of 0-5 V, and two capable of 0-15 V. To power the electronic components for the lab, we will need a +12 V supply, a –12 V supply, and a ground connection. We will use the two 0-15 V outputs in series and adjust them to output approximately 12 V each.

1. Switch the power supply to "series".
2. Adjust the "A" power to ~ 12 V (the "B" output will match the "A" output).
3. Connect a red cable to the "+" terminal for the "A" power output.
4. Connect a black cable to the "–" terminal for the "B" power output.

In this configuration, the negative side of the "B" output is a –12 V source. The positive side of the "A" output becomes the +12 V source. The positive side of the "B" output and the negative side of the "A" output are the ground.

Incorrectly connecting an op-amp to the power supply will destroy the op-amp. Make sure that the op-amp is connected with correct polarity.

Part 2: Preparing the Breadboard

1. Connect the +12 V supply to one of the strips in the A region.
2. The other strip in the A region should be tied to ground.
3. In the D region, one of the strips should be the –12 V bus.
4. The other D region strip should be used as a ground bus.

Your breadboard is now configured to power the circuits for this lab.

Part 3: Generate a Simulated Signal

Use the function generator to create a sine wave with the following characteristics:

- 50 Hz
- amplitude of 80 mV
- offset (bias) of approximately 800 mV

For such a small signal, the Attenuator button on the function generator must be depressed. This signal will be treated as the output of a sensor. We will condition this "sensor output" with a signal conditioning circuit.

Part 4: Modify Existing VI for Multiple Inputs

In this lab, we will build on the skills that were developed in [Introduction to Benchtop Equipment and Data Acquisition](#) and [Temperature Measurement and First-Order Dynamic Response](#). It is assumed that the student understands the concepts of a Front Panel, a Block Diagram, the Control Palette, and the Functions Palette. It is also assumed that the student can search the control and functions palette for necessary features. The student should be familiar with configuration options for voltage measurements using the DAQ Assistant.

The VI that you developed for [Temperature Measurement and First-Order Dynamic Response](#) can be easily modified to examine circuit inputs and outputs.

1. Open the VI named Lab3 that you saved during [Temperature Measurement and First-Order Dynamic Response](#).
2. Open the DAQ Assistant Configuration Box.

   1. Delete the Voltage channel.
   2. Add a voltage channel for module 1, analog input 0.
   3. Add a voltage channel for module 1, analog input 1.
   4. For the Terminal Configuration, select Differential.
   5. For Acquisition Mode, select Continuous.

3. Click File>>Save As…>>Substitute Copy for Original to save your VI with a new name.
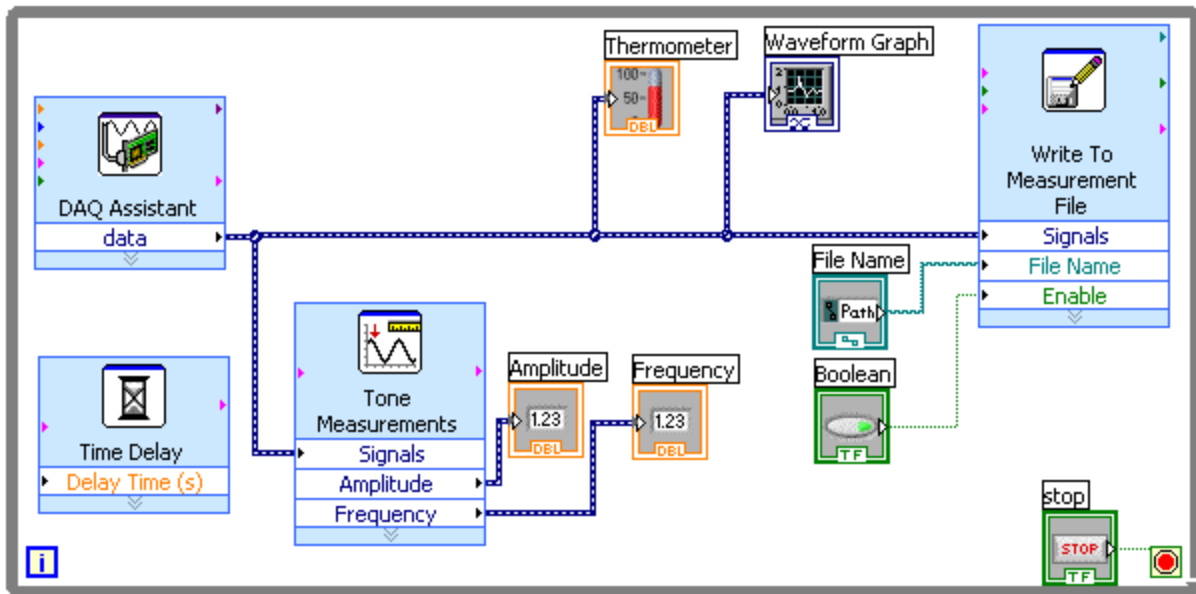
Modified Block Diagram

Part 5: Op-Amp Circuits

Before building a circuit to condition the signal, you will build a couple of simple op-amp circuits to get a feel for what to do.
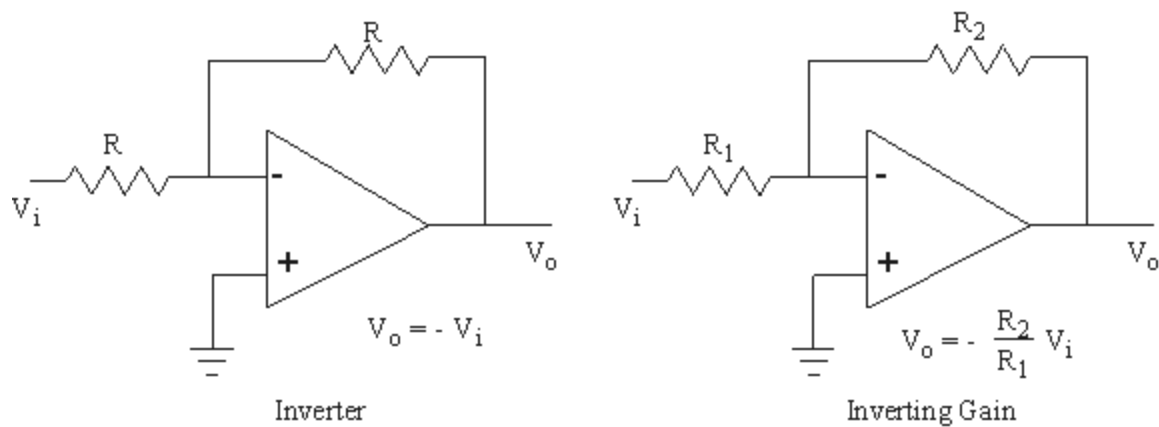
5.1 Inverting Circuit

The inverter is one of the simplest op-amp circuits. This circuit simply changes the sign of a signal. The schematic diagram of an inverter is shown in Figure 6.

1. Build the inverter using two 10 k ohm resistors and an LM741 op-amp.
2. Input the signal from the function generator to the circuit.
3. Branch the input cable with a BNC T-connector. Connect this input to Channel 0 on Module 1 of the SCXI.

4. Connect the output signal from the circuit to the Channel 1 on module 1 of the SCXI.
5. When you run the VI, you may need to adjust the scales on the X and Y axes.

You should see that the output is an inverted version of the input.

Inverter and Inverting Gain Circuits

5.2 Inverting Gain Circuit

The inverting gain circuit changes the sign of the signal and multiplies the amplitude. Increasing the amplitude of a signal can be very important in distinguishing subtle signal characteristics. Quantization errors result when a signal characteristic is too small to be detected by a measurement system. (You will learn more about quantization errors in lecture, and on pages 249-250 in your text.)

1. Replace the feedback resistor (R2 as shown in Figure 5) in your inverter circuit with a 100 k ohm resistor. This will cause the input signal to be inverted and amplified by a factor of ten.
2. Run the VI again.

With the amplification, the input offset of 800 mV has increased to an 8 V output.

Note: Using the data acquisition card to view the signal, you may saturate the ADC with the amplified signal. All you will be able to see is a flat line near +5 V or –5 V.

Part 6: Bias Correction

For the amplified signal to be read correctly, the 800 mV bias must be removed. A two step process will be used to remove the bias:

- A Trim Potentiometer will be used to step down a 12V DC signal to 800 mV.
- A Differential Amplifier circuit will be built to subtract this stepped down voltage from the sine wave.

The result will be an 80 mV sine wave with zero mean.

6.1 Trim Potentiometer Adjustment

Use the 10 k ohm trim potentiometer provided to build a voltage divider as shown below.

1. Connect one end lead of the potentiometer to the +12 V bus.
2. Connect the other end lead to the –12 V bus.
3. Connect a voltmeter across the wiper lead and ground to measure the output.
4. Adjust the potentiometer resistance until the voltage on the wiper lead is 800 mV.

```
+12 V  o
          wiper
                 o V_out
- 12 V  o
```
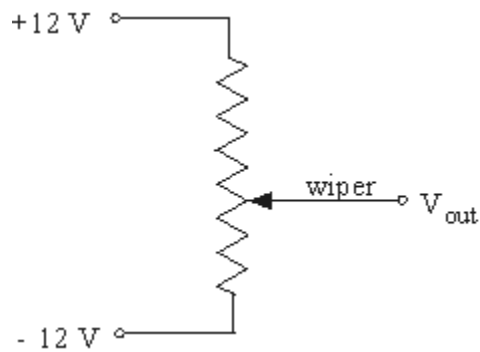
Voltage divider circuit

6.2 Differential amplifier

You will build a differential amplification circuit to subtract 800 mV from the sensor signal. A differential amplifier is shown in Figure 8 below.

1. Use four 10 k ohm resistors to subtract the voltage divider output from the sensor signal. It is unlikely that the resulting output signal will be a zero-mean sine wave as desired.
2. Adjust the screw on the potentiometer to "trim" the offset of the sine wave to zero.
3. Amplify the difference between the signals by replacing the R2 resistors so that R2 > R1. Replace the R2 resistors with 100 k ohm resistors.

The resultant signal should be an 800 mV sine wave with minimal offset.

Using the inverting gain circuit shown in Figure 8, amplify the output of the differential amplifier circuit by an additional factor of ten. This can be done with another op-amp and a 10 k ohm resistor and a 30 k ohm resistor. The resultant output signal should have a mean of zero and amplitude of ~2.4 V. Adjust the potentiometer to remove any offset.

$$V_o = - \frac{R_2}{R_1} (V_1 - V_2)$$
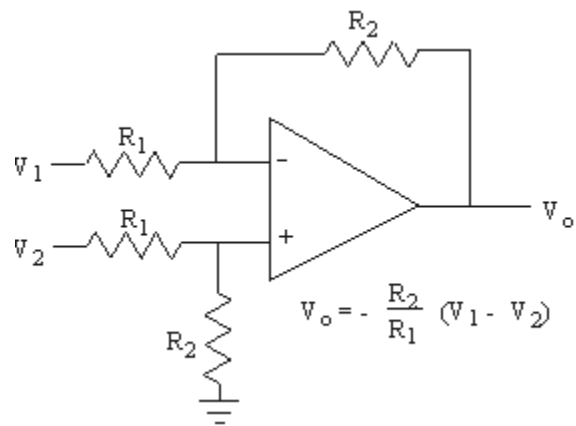
Differential Amplifier Circuit

Part 7: Low-Pass Filtering

A low pass filter can be used to attenuate high-frequency noise in an analog signal and to minimize the portion of the signal that will be aliased. Later in this lab, you will see a demonstration of the damaging effects of aliasing.


$$\text{static gain} = \frac{R_2}{R_1}$$

$$\text{cut-off frequency (Hz)} = \frac{1}{2 \pi R_2 C_2}$$
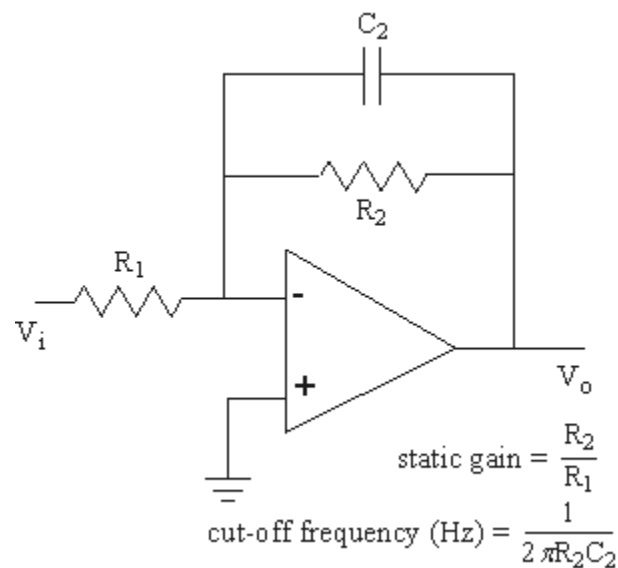
First-Order Low-Pass Filter

7.1 Build an Active Low-Pass Filter

1. Build the first-order filter shown in Figure 9 with R1 = R2 = 10 k ohm and C2 = 0.1 micro F. First order filters are so-called because their dynamics are modeled by first-order differential equations.
2. Connect the output of the circuit to Channel 1 on your DAQ system.
3. The filter's time constant is equal to R2C2. Calculate values for the cut-off frequency and the time constant.

7.3 Modifying Existing VI to Measure Magnitude Ratio

By measuring the magnitude of the input and output of a filter, you can determine the how much the filter attenuates the signal.
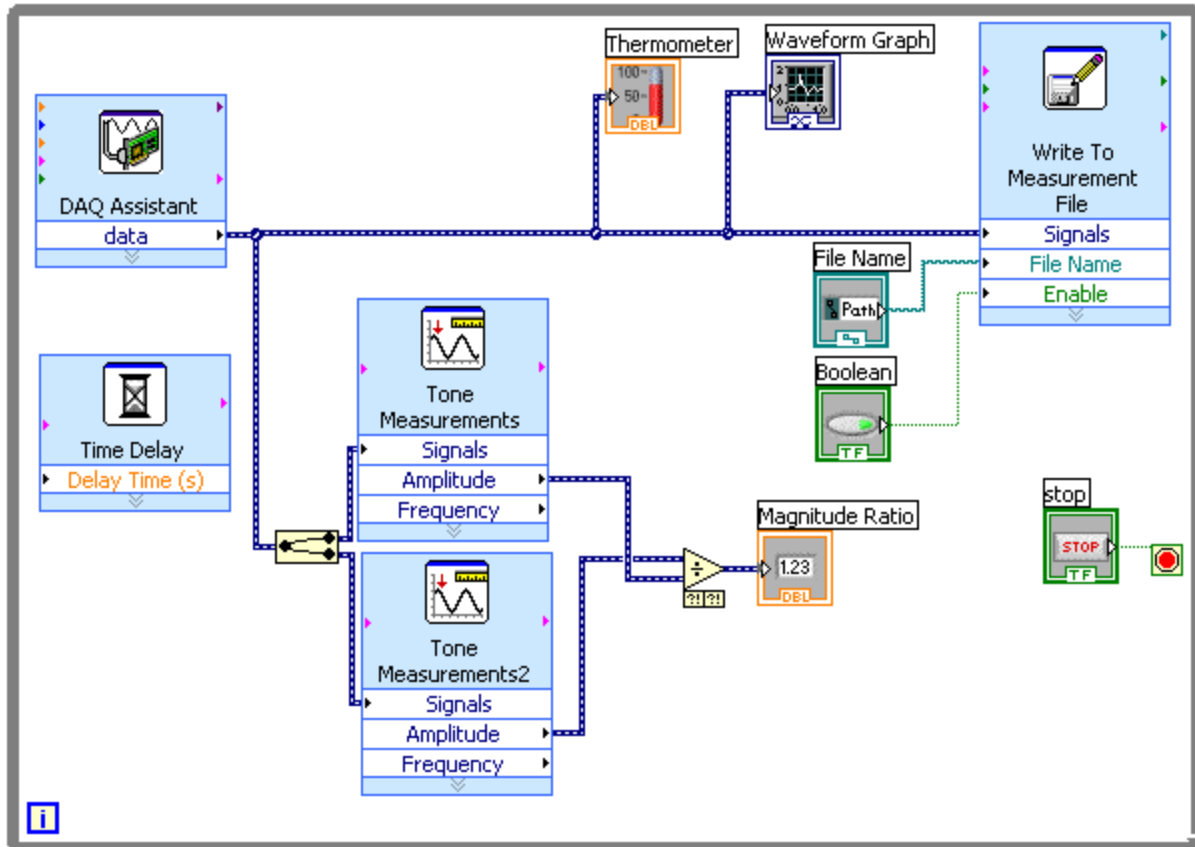
Completed Block Diagram

1. Delete the Amplitude and Frequency constants from the Block Diagram.
2. Hold Ctrl as you drag the Tone Measurements icon to make a copy.
3. Place a Split Signals icon to the left of the Tone icons.
4. Place a Divide function to the right of the Tone icons.
5. Create a numeric indicator at the x/y output terminal of the Divide function.
6. Rename the numeric indicator Magnitude Ratio. (The Magnitude Ratio represents the output amplitude divided by the input amplitude.)
7. Wire the Block Diagram as shown in Figure 10.
8. Save the VI.

7.2 Testing a Filter

We expect that the filter will allow frequencies below the cut-off frequency to "pass", and will attenuate signals at higher frequencies. The Magnitude Ratio will be recorded as the output magnitude divided by the input magnitude.

1. Connect the signal going into your filter to Channel 0 of module 1 of the SCXI.
2. Connect the signal coming out of your filter to Channel 1 of module 1 of the SCXI.
3. Input a 1 V sine wave with zero offset into the low-pass filter circuit.
4. Starting with a frequency of about 10 Hz, slowly increase the frequency of the input signal. What happens to the output signal?
5. Increase frequencies to 2 kHz.
6. Using the table below, determine the magnitude ratio at several frequencies.

Table 1: Magnitude Ratio Data for First-Order Low-Pass Filter (Active)

frequency (Hz)	Input magnitude (V)	Output magnitude (V)	Magnitude ratio
10	1V		
18	1V		
32	1V		
58	1V		
110	1V		

190	1V		
340	1V		
620	1V		
1100	1V		
2000	1V		

1. Using Excel, plot the following two sets of data on a single log-log chart.

1. 1. Calculated magnitude ratio vs. frequency.
   2. Measured magnitude ratio vs. frequency

Do the magnitude ratio and phase difference between the input and output behave as you would expect?

1. Input a 30 Hz square wave into the low-pass filter. Record your observations.
2. Measure the time constant from the oscilloscope screen.

Does it correlate with your calculated values based on resistor and capacitor values?

7.3 Build a Passive Low-Pass Filter

The filter described in step 6.1 is an active filter, meaning it requires an external power source. Since it uses an op-amp, it must have a ±12 V power supply. Active filters can have a static gain greater than one. Active filters also have low output impedance, which means that they can pass up to 10 mA of current without any effects on the operation of the filter.

A passive low-pass filter does not require an external power source. A passive filter can be constructed with simply a resistor and capacitor as

shown in Figure 11.

1. Build a filter using a 10 k ohm resistor and a 0.1 micro F capacitor.
2. Repeat section 6.2 (steps 4-11). Record your results in your lab book. You may use Table 2 to organize your data.

$$V_i \dashrightarrow R \dashrightarrow V_o$$

$$\text{static gain} = 1$$
$$\text{cut-off frequency (Hz)} = \frac{1}{2\pi RC}$$
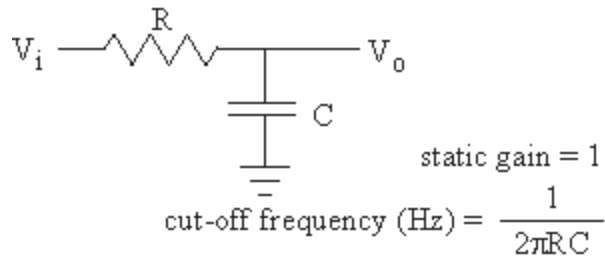
Passive First-order Low-pass filter

Table 2: Magnitude Ratio Data for First-order Low-pass Filter (Passive)

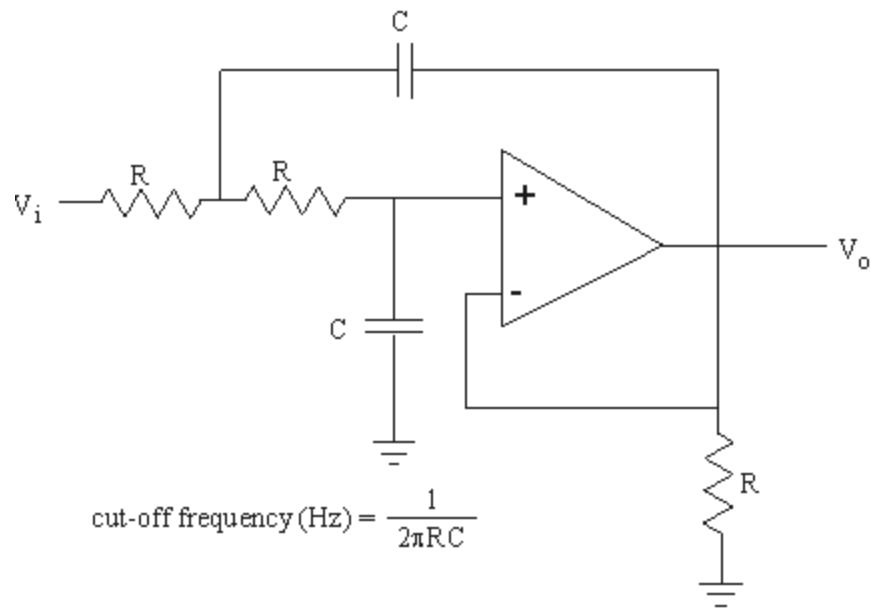| frequency (Hz)	Input magnitude (V)	Output magnitude (V)	Magnitude ratio
10	1V		
18	1V		
32	1V		
58	1V		

110	1V		
190	1V		
340	1V		
620	1V		
1100	1V		
2000	1V		

7.4 Build a Second-Order Butterworth filter

Figure 12 below shows a second-order Butterworth low-pass filter. A second-order filter is superior to a first-order filter in many respects, as we will investigate.

1. Build the Butterworth filter circuit using R = 10 k ohm and C = 0.1 micro F.
2. Perform the sine wave and square-wave input tests from step 6.2. Record your results in your lab book. You may use Table 3 to organize your data.

How is the response of this filter different from that of the first-order filter?

$$\text{cut-off frequency (Hz)} = \frac{1}{2\pi RC}$$

Second-order Butterworth Low-pass Filter

Table 3: Magnitude Ratio Data for Second-Order Low-Pass Filter

frequency (Hz)	Input magnitude (V)	Output magnitude (V)	Magnitude ratio
10	1V		
18	1V		
32	1V		
58	1V		
110	1V		

190	1V		
340	1V		
620	1V		
1100	1V		
2000	1V		

Part 8: Aliasing

Aliasing is a generally undesired phenomenon that results when periodic signals are sampled too slowly. Aliasing causes high frequency periodic signals to be recorded as signals consisting of lower frequencies, as will be apparent in the following steps.

8.1 Non-Filtered Signal

You will cause a signal to be aliased as it is converted from an analog to a digital signal.

1. On the block diagram, create Numeric Indicators to display Frequency and Amplitude.
2. Wire the Numeric Indicators to the Tone Measurements icon for the second channel.
3. Change the sample rate of the DAQ Assistant to 1000 S/sec (samples per second).
4. Calculate the Nyquist Frequency for this sample rate.
5. Connect the output of the Function Generator directly to the second input channel on the SCXI.
6. Adjust the function generator to create a sine wave with a frequency of 50 Hz and amplitude of 1 V.

7. Slowly increase the frequency of the sine wave until you reach 400 Hz. Observe what happens to the sampled version of the sine wave.
8. Increase the frequency to 500 Hz. What is the measured frequency and amplitude?
9. Why doesn't LabVIEW display a "true" image of the wave form?
10. Continue to increase the frequency of the sine wave to 1000 Hz.
11. Describe what happens to the amplitude and the frequency of the sampled signal as the frequency increases.
12. Why is the frequency displayed in LabVIEW different from the frequency supplied by the function generator? How are the two frequencies related?

8.2 Anti-Aliasing Filter

Low pass filters are frequently employed to minimize the portion of a signal that is aliased.

1. Input the signal from the function generator to the second-order Butterworth filter from Section 6.2.
2. Connect the output of the filter to channel 2 of the SCXI.
3. Set the sample rate to 1 kS/sec (as before).
4. When used in this way, the filter is called an anti-aliasing filter. Comment on what you observe. How are your observations different from those in step 10 without the filter?

Bonus

Estimate the damping ratio of the second-order Butterworth filter through experimentation. To do this, you will probably need to take more measurements near the expected cut-off frequency of the filter (which is near the natural frequency). Support your result with plots and discussion as necessary.

Lab Report

For this lab, you will write up the Introduction and Objective sections of a full formal report. You will be provided handouts with further instruction regarding what is expected.

Strain Gages and Force Measurement

Introduction

In this lab, you will learn about strain gages and the Wheatstone bridge circuit. You will see how they can be used for strain and force measurement. You will modify an existing program to measure the dynamic characteristics of a second-order system.

Teaching Objectives

- Gain practical experience with resistance strain-measurement techniques.
- Learn about the Wheatstone bridge and how it is used in strain measurement.
- Use a beam instrumented with strain gages as a force measurement device.
- Use strain gages to measure the natural frequency and damping in a beam.
- Design a force transducer for measuring thrust from a model rocket motor.

# Preparatory Reading:

Figliola and Beasley

Strain Measurement: pp. 425–446

Procedure

Part 1: Strain Gages and the Wheatstone Bridge

The metal foil strain gages used in this lab are resistors with a nominal (unstrained) resistance of 120 ohms. As they are put in tension, their resistance increases; as they are compressed, their resistance decreases. The Wheatstone bridge provides a way to convert these changes in resistance to

changes in voltage, which are easy to work with. These voltages can be conditioned, transmitted, or stored digitally.
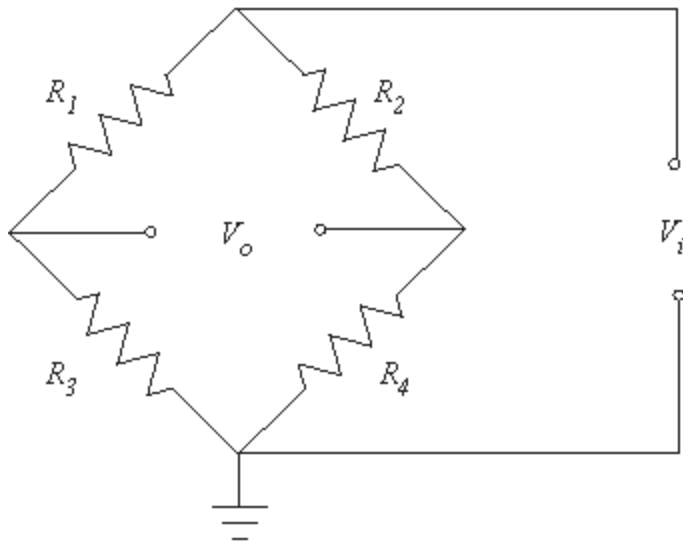
Wheatstone Bridge Circuit

Figure 1 shows a Wheatstone bridge configuration.

- Four resistors are connected in an end-to-end fashion.
- The input or excitation voltage is connected to the bridge between top and bottom nodes of the circuit.
- The output is the difference between the voltage at the left node and the voltage at the right node.
- An excitation voltage is required to convert the change in resistance (in the legs of the bridge) to a change in voltage at the output of the bridge.

For the bridge shown, the output voltage is expressed as

$$V_o = \left(\frac{R_3}{R_1 + R_3} - \frac{R_4}{R_2 + R_4} \right) V_i$$

equation (1)

When building a Wheatstone bridge with strain gages, all four resistors have the same nominal value. Bridges can be built in the following configurations:

- Quarter Bridge-One strain gage and three fixed resistors
- Half Bridge- Two strain gages and two fixed resistors
- Full Bridge- Four strain gages
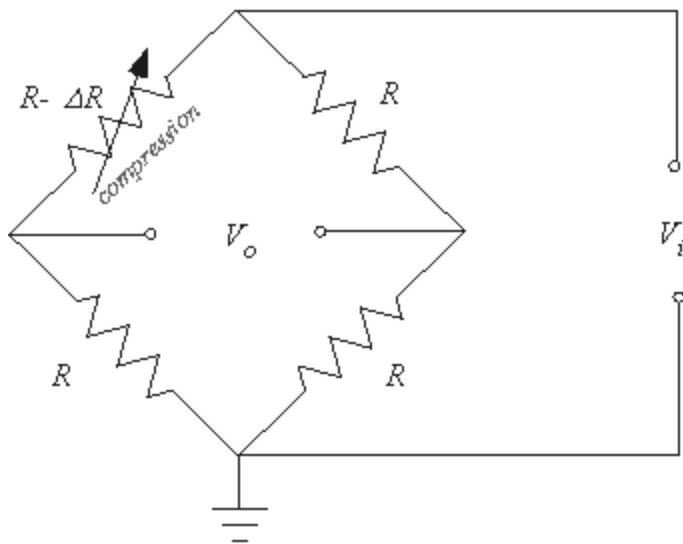
Quarter Bridge Configuration

Quarter Bridges

Figure 3 illustrates a quarter bridge configuration. The quarter bridge has one active leg, i.e., one leg with a changing resistance. From equation (1) above we can derive an expression for the output voltage as a function of the resistance change ΔR:

$$V_o = \left(\frac{R_3}{R_1 + R_3} - \frac{R_4}{R_2 + R_4} \right) V_i = \left(\frac{R}{(R - \Delta R) + R} - \frac{R}{R + R} \right) V_i$$

$$= \left(\frac{R}{2R - \Delta R} - \frac{1}{2} \right) V_i = \left(\frac{2R - (2R - \Delta R)}{2(2R - \Delta R)} \right) V_i$$

$$= \left(\frac{\Delta R / R}{2(2 - \Delta R / R)} \right) V_i$$

$$\approx \frac{\Delta R}{4R} V_i.$$

equation (2)

Half and Full Bridges

Figure 5 and Figure 6 show half-bridge and full-bridge configurations respectively.

- Half bridge: two active legs, one in tension and one in compression. These legs are adjacent legs in the bridge.
- Full bridge: four active legs, two in tension and two in compression. The gages in tension are on opposite legs of the bridge.
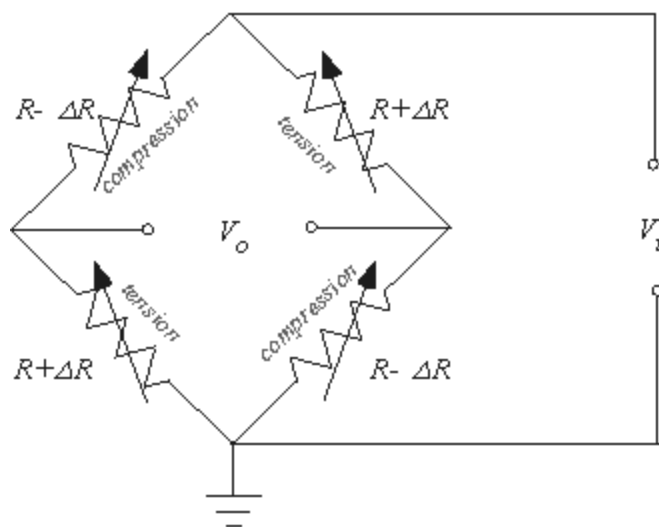
Using equation 1 and Figure 1 as a guide, derive expressions for the output voltage of the half-bridge and full-bridge circuits.

Thought Question

A half bridge could be made with two gages in tension on opposite legs.
When would this be useful? What would be the main problem with doing
this?
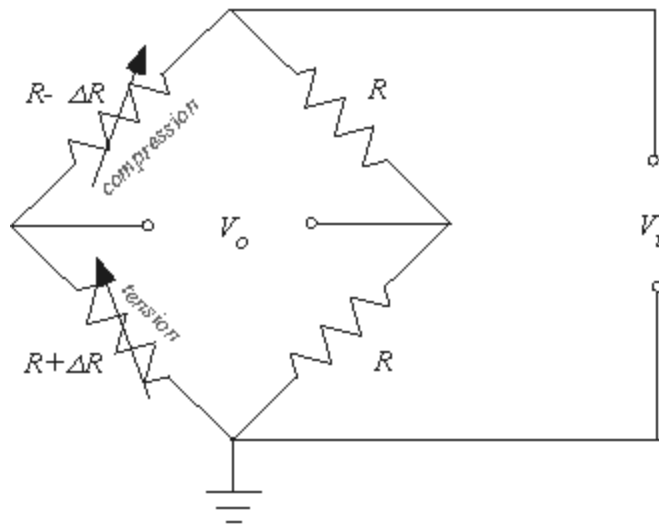
Half Bridge Configuration

Full Bridge Configuration

Part 2: Calibration of the Strain-gaged Cantilever Beam

Your TA will provide an aluminum beam instrumented with strain gages.

1. Clamp the beam securely to the edge of the lab table with the gaged portion of the beam cantilevered beyond the edge of the bench.

2. Using the junction box provided, connect a single strain gage in a quarter bridge configuration. (Refer to Figure 3 as a guide.)

3. Connect the junction box output cable to Channel 2 of the SCXI-1520 strain gage module. The SCXI-1520 provides an excitation voltage to the bridge and amplifies the output voltage from the bridge to ranges that are easily observable or acceptable for data acquisition. The excitation voltage Vi is 5 V and the gain is 500. When connected to the signal conditioning board, the amplified bridge output can be read by the data acquisition software.

4. Open the "Lab4" VI that you created in [Temperature Measurement and First-Order Dynamic Response](#).

5. Hang weights from the end of the beam.

6. Record the voltage measured with LabVIEW in your lab book.

7. Repeat steps 5 and 6 for several different weights.

8. In Excel, plot the voltage (input) versus weight (output).

9. Find the best fit linear relationship for the data. The resulting equation can be used to calibrate the voltage output of the strain gages.

10. Apply several loads not used for calibration to test the validity of the linear curve fit.

Part 3: Comparison of Theoretical Strain to Measured Strain

In this step, we will compare a theoretically based estimate of strain for a given load to that which was measured earlier. First we will determine the strain corresponding to the voltage measurements of step 2.

1. Measure the excitation voltage for bridge circuit, Vi.

The circuit analyses from step one showed that the bridge output Vo is related to the excitation voltage by the following relationship

$$V_o = K \frac{\Delta R}{R} V_i$$

    equatio
     n (2)

where K equals 1/4 for a quarter bridge, 1/2 for a half bridge, and 1 for a full bridge. The voltage measured in LabVIEW is related to the bridge output by

$$V_{display} = K_{amp} V_o + V_{offset}$$

    equation (3)

where Kamp is the gain of the signal conditioning board. Because the bridge resistances are not balanced exactly and the weight of the beam itself produces some strain, you will observe a nonzero output voltage when there is no load applied.

2. Measure the output voltage with no load. Call this voltage Voffset.

3. To determine the strain induced by the applied loads, measure the changes in the display voltages relative to this offset voltage (Vdisplay–Voffset).

For a strain gage, the gage factor is defined as

$$GF = \frac{1}{\varepsilon} \frac{\Delta R}{R}$$

equatio
n (4)

where epsilon is the strain experienced by the gage. The gages used in this lab have a gage factor of 2.12±0.8%.

4. Derive an expression for the strain in the beam using equations (2) through (4). Your empirical strain estimate should be in the range of 0 to 2000 microstrains.

A theoretical estimate of the strain can be obtained by drawing on topics from CEEn 203. The stress on the surface of a beam in bending is given by

$$\sigma = \frac{My}{I}$$

equatio
n (5)

where M is the applied moment at the location of interest, y is the distance from the neutral axis (in this case, the half height of the beam cross

section), I is the area moment of inertia of the cross section with respect to the neutral axis. Recall that for a rectangular cross section,

$$I = \frac{1}{12} bh^3$$

equatio
n (6)

Recall also that stress and strain are related by Young's modulus:

$$\sigma = E \varepsilon$$

equatio
n (7)

For aluminum, E = 10.4x10^6 psi.

5. Using equations (5) through (7), estimate the strain where the gages are bonded to the beam. How does the theoretically obtained strain compare to the value determined from measurements? If they are different, what are some possible reasons?

Part 4: Experiments with Half and Full Bridges

Repeat parts 2 and 3 for the half bridge and the full bridge configurations.

Part 5: Measurement of an Unknown Load

Based upon the calibration determined in step 2, use your beam to determine the weight of an arbitrary object. You may use the quarter bridge, the half bridge, or full bridge configuration for this test. Measure the actual weight using a precision scale. How does the weight determined with your beam compare to the object's true weight? How certain is your measurement? What are some possible sources of uncertainty?

Part 6: Sensitivity to Extraneous Loads and Temperature Variations

Strain gages are often used as transducers to measure force in a structure or mechanical device. A perfect sensor is sensitive only to changes in the variable of interest and is completely insensitive to changes in extraneous parameters. (Such a sensor is impossible to create.) In practice, every effort should be made to minimize sensitivity to extraneous variables. In the case of the cantilever-beam scale, the objective is to measure the weight of objects or alternatively, to measure forces in the vertical plane. As such, a good design will be much less sensitive to loads in other directions (axial, lateral, torsional, etc.). Also a good design will not be sensitive to strains induced by other means, such as thermal expansion.

Your TA will supervise in the following two demonstrations.

6.1 Lateral Sensitivity

Use the instrumented beam of square cross section provided by your TA.

1. Clamp the beam in a cantilever fashion so that it will measure vertically applied loads.

2. Using your hand, gently apply lateral and axial loads to the beam. Do your best to not apply vertical bending loads to the beam.

3. Compare the sensitivity of the quarter bridge to that of either the half bridge or full bridge to these extraneous loads. Which configuration seems to work the best overall? Explain why this is so. Which extraneous loads are "rejected" by which configurations? Explain why. (Note: The steps of this paragraph are more easily said than done. The point is that the quarter bridge is more sensitive to off-axis loads than either the half bridge or full bridge. If you have difficulty demonstrating this, don't stress out.)

6.2 Temperature Sensitivity

Using a quarter bridge configuration, apply a load to the beam and measure the voltage output.

Apply heat uniformly to the beam (140 F max) near the gages and observe any changes in the voltage output. Do not apply heat to the gages directly! Why does the voltage (which corresponds to the measured strain) change? Repeat the procedure using a full-bridge configuration. Compare the sensitivity of the quarter bridge and full bridge configurations to temperature variations. How would you expect the half bridge to behave?

Part 7: Dynamic Characteristics

Up to this part of the lab, you have examined the static characteristics of the strain-gage bridge. Now you will modify the VI that you developed for [Temperature Measurement and First-Order Dynamic Response](#) to measure the dynamic characteristics of a signal from the strain-gage bridge. You will excite the dynamics of the cantilever beam by plucking it. You will use your VI to display the response of the vibrating beam.

7.1 Differential Model

The first mode of vibration of the cantilever beam can be modeled using a simple mass-spring-damper model. This model results in a second-order

differential equation that describes the dynamics of the system. You will observe that the response of the strain –gages to an initial deflection is a damped sinusoid. This is the expected response for a second-order system. Using acquired data; you will compute and display values for the damping ratio and the natural frequency of the first vibrational mode of the beam.

7.2 Damping Ratio and Damped Natural Frequency

Figure 5 shows the response of a second-order system to an initial condition. This response plot will be used to define the damping ratio and the natural frequency for this system. From the amplitudes x1and xn, the damping ratio can be calculated using the following expression:

$$\zeta = \frac{\frac{1}{n-1}\ln\left(\frac{x_1}{x_n}\right)}{\sqrt{4\pi^2 + \left[\frac{1}{n-1}\ln\left(\frac{x_1}{x_n}\right)\right]^2}}$$

equation (8)

The damped natural frequency can be determined by measuring the period of the damped oscillations. An accurate measurement of the period T is obtained by considering several periods:
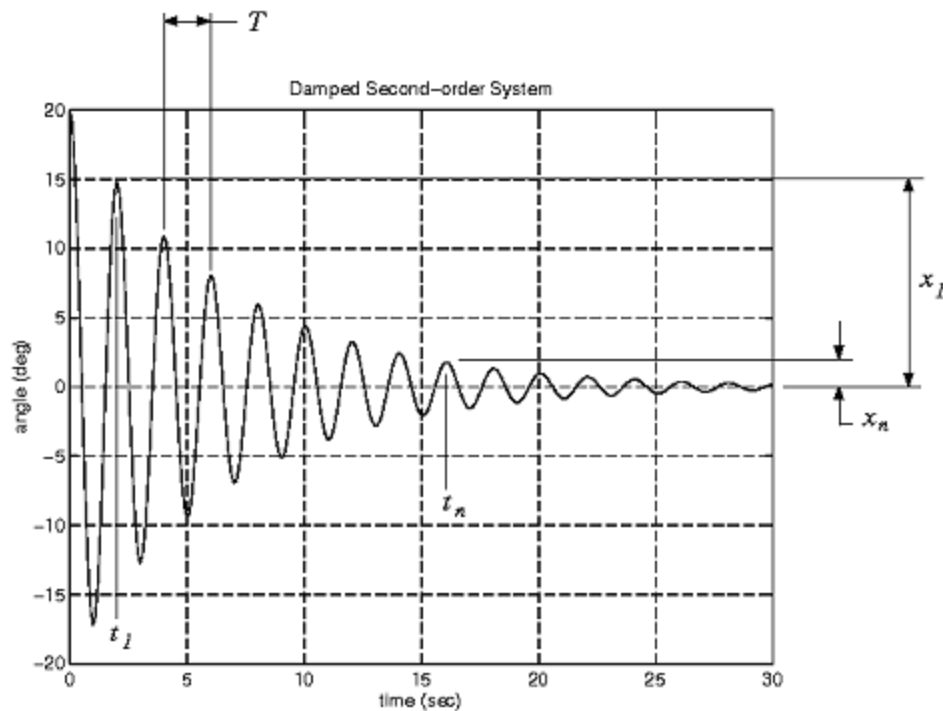
$$T = \frac{t_n - t_1}{n - 1}$$

equatio
n (9)

$$\omega_d = \frac{2\pi}{T}$$

equatio
n (10)

Once and are known, the natural frequency of oscillation can be calculated by

$$\omega_n = \frac{\omega_d}{\sqrt{1 - \zeta^2}}$$

equatio
n (11)

Second-order system Response

7.3 Programming Exercise

You will modify your VI to automate the calculation of and for the cantilever beam.

1. Your TA will direct you to a subVI named "Damping Ratio" that is has been started for you.

2. To place the subVI on your block diagram, open the functions palette and select Select a VI…

3. Connect the Data wire from the DAQ Assistant to the Measured Data input of the Damping Ratio subVI.

4. Open the block diagram by double-clicking the subVI icon. (The block diagram is shown in Figure 18.)

5. You will need to modify the block diagram for the VI to work. Refer to equations 8-11 as you design a block diagram that will calculate the Damping Ratio of your signal.
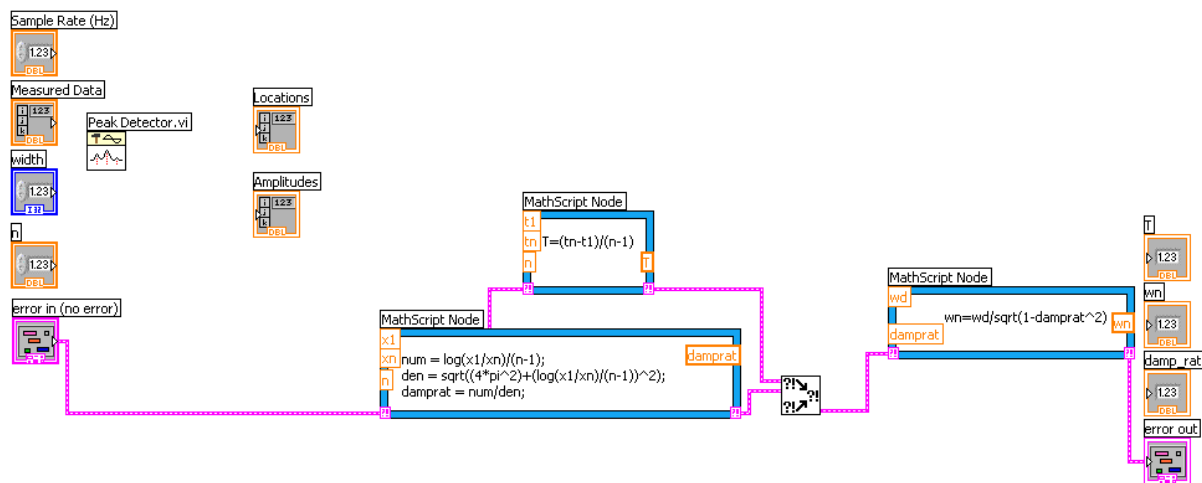
Damping Ratio SubVI Block Diagram

6. You will notice that the necessary inputs for the subVI are placed along the left of the block diagram. The outputs are placed along the right of the block diagram. The Peak Detector VI is used to locate the peaks (or valleys) of the signal. The locations of the peaks are given as indexed data points. For example, if the first peak occurs in between the sixth and seventh data points, the location will be 6.5. The time at which this peak occurs can be calculated using the sample rate.

7. For more information about the Peak Detector or other VIs, press Ctrl+H to activate the context help window.

(Hint: you may want to use the index array function to pull values such as x1, xn, t1 and tn out of the amplitude and location arrays.)

This exercise is meant to give you a feel for the possibilities for data acquisition, processing, and display using computer software tools. Be sure to save a copy of your data for plotting. You will need it for your writing assignment.

Part 8: Design of a Force Transducer

For your course project, you will design a force transducer for measuring the thrust of a model rocket motor. This exercise is intended to help you initiate the design process. You may assume that the maximum thrust that will be sensed by your transducer is 5 lb.

Cantilever-Beam Force Transducer

Assume that you will use a cantilever-beam configuration similar to what was used in this lab. A full strain-gage bridge will be used. The width of two gages side-by-side is 1/2 inch. A good design will result in 2000 microstrains under the maximum load and will have a natural frequency above 100 Hz. Meeting both of these objectives might not be possible with a cantilever-beam design. You are to determine the thickness of the cross section where the gages are to be mounted and the distance from where the motor is mounted to the center of the gages (i.e., the length of the cantilever).

Spreadsheet Analysis

Develop a spreadsheet to perform your calculations. A spreadsheet will simplify the evaluation of different designs. This analysis will be a good starting point for the analysis of your project. Note: Your project will involve different loads and different dimensions than the transducer used in this lab. The calculations, however, are the same.

Lab Report

For this lab you will write the Results and Discussion of Results sections of a full report. As you perform the lab, think about what data should be saved or recorded for presentation and why these data are important. For some of your data, tabulation is sufficient (e.g., calculated strain vs. measured strain). Other data should be recorded using LabVIEW (e.g., dynamic response of strain when the beam is plucked). Make sure that you include your thoughts about the results you obtained and why they are important. Discuss their agreement with your expectations.